

NASA/TM-2008-215318



Focused Assessment of State-of-the-Art CFD Capabilities for Prediction of Subsonic Fixed Wing Aircraft Aerodynamics

*Christopher L. Rumsey and Richard A. Wahls
Langley Research Center, Hampton, Virginia*

June 2008

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the lead center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.
- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results ... even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA STI Help Desk at (301) 621-0134
- Phone the NASA STI Help Desk at (301) 621-0390
- Write to:
NASA STI Help Desk
NASA Center for AeroSpace Information
7115 Standard Drive
Hanover, MD 21076-1320

NASA/TM-2008-215318



Focused Assessment of State-of-the-Art CFD Capabilities for Prediction of Subsonic Fixed Wing Aircraft Aerodynamics

*Christopher L. Rumsey and Richard A. Wahls
Langley Research Center, Hampton, Virginia*

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23681-2199

June 2008

Available from:

NASA Center for Aerospace Information (CASI)
7115 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 605-6000

Focused Assessment of State-of-the-Art CFD Capabilities for Prediction of Subsonic Fixed Wing Aircraft Aerodynamics

Christopher L. Rumsey*

Richard A. Wahls†

NASA Langley Research Center, Hampton, VA 23681-2199, USA

Several recent workshops and studies are used to make an assessment of the current status of CFD for subsonic fixed wing aerodynamics. Uncertainty quantification plays a significant role in the assessment, so terms associated with verification and validation are given and some methodology and research areas are highlighted. For high-subsonic-speed cruise through buffet onset, the series of drag prediction workshops and NASA/Boeing buffet onset studies are described. For low-speed flow control for high lift, a circulation control workshop and a synthetic jet flow control workshop are described. Along with a few specific recommendations, gaps and needs identified through the workshops and studies are used to develop a list of broad recommendations to improve CFD capabilities and processes for this discipline in the future.

Nomenclature

AIAA = American Institute of Aeronautics and Astronautics

AR = wing aspect ratio

$B/2$ = wing semi-span

BB = Baldwin-Barth turbulence model

CC = circulation control

CFD = computational fluid dynamics

C_L = lift coefficient

C_D = drag coefficient

C_{dp} = pressure drag coefficient

C_p = surface pressure coefficient

C_μ = jet momentum coefficient

c = chord length

DES = detached eddy simulation

DNS = direct numerical simulation

DoD = Department of Defense

DPW = Drag Prediction Workshop

EASM, EASM-ko = explicit algebraic stress turbulence model

ERCOTAC = European Research Community On Flow, Turbulence and Combustion

L = physical length scale (typically airfoil chord or wing mean aerodynamic chord)

LES = large-eddy simulation

\dot{m} = mass flow rate

NASA = National Aeronautics and Space Administration

NPTS = number of total grid points

N-S = Navier-Stokes

RANS = Reynolds-Averaged Navier-Stokes

Re = Reynolds number

SA = Spalart-Allmaras turbulence model

SARC = Spalart-Allmaras with Rotation and Curvature turbulence model

*Senior Research Scientist, Computational Aerosciences Branch, Mail Stop 128, c.l.rumsey@nasa.gov.

†Senior Research Scientist, Configuration Aerodynamics Branch, Mail Stop 267, richard.a.wahls@nasa.gov.

SST = Menter shear stress transport turbulence model

U_∞ = freestream velocity

u = velocity

u_j = jet velocity

uv = turbulent shear stress

V&V = Verification and Validation

WB = wing-body

WBNP = wing-body-nacelle-pylon

X_B = bubble length

x, y, z = coordinate directions

α = angle-of-attack

ρ_∞ = freestream density

I. Introduction

Efficient flight has at its foundation the understanding and prediction of fluid flow around complex geometries - aerodynamics. Aerodynamicists strive to improve the efficiency of aircraft configurations via the research, development, and integration of advanced aerodynamic technology. Validated capabilities to predict the performance of such technologies are needed at relevant conditions to enable practical implementation on a aircraft configuration. The combination of advanced technology and high confidence predictive tools gives end users – the aircraft designers – a larger, viable design space for developing future vehicles. The underlying challenge is high-confidence, pre-flight prediction of complex flow phenomena at flight conditions.

The Subsonic Fixed Wing Project within NASA’s Fundamental Aeronautics Program has the goal of dramatically improving the noise, emissions, and performance characteristics of future aircraft. Advanced concept research is guided by projected vehicle capabilities to meet future requirements, such as increasingly stringent environmental impact constraints. To meet this goal, the project has as one of its objectives the development and validation of improved prediction and analysis tools to enable reduced uncertainty in the design process. Today, and into the foreseeable future, these tools include empirical, computational, and experimental methods working synergistically to predict flight characteristics prior to first flight in an efficient and accurate manner. High-confidence prediction tools are needed throughout the full envelope as aerodynamics drives vehicle outer mold line definition for performance and provides input to many other disciplines in the design process, such as loads input for controls and structures disciplines. Aerodynamics research in this project generally focuses on technologies for efficient cruise and low-speed high-lift performance, targeting improvements in cruise Mach number times lift-over-drag (ML/D), and low-speed maximum lift coefficient ($C_{L,max}$). Aerodynamic technologies include both active and passive concepts, from a component to configuration level. Cross-cutting the technologies are the predictive challenges of flow separation onset and progression along with laminar flow, boundary-layer transition, and turbulence.

Predicting characteristics across the flight envelope prior to flight has always been challenging. Issues related to Reynolds number scaling in wind tunnels and viscous sensitive phenomena in computations often make it difficult to put a high confidence level on the predictions. As discussed in Wahls¹ and other references cited therein, there have been many examples documented for which flight performance was unexpectedly either worse or better than predicted beforehand. These discrepancies make management of the aircraft design and development process very difficult, and can cost companies a lot of money. Although capabilities for specific flight regimes have been improving (such as computational tools for attached turbulent boundary layer flow), improved prediction methods are still generally needed that work consistently for a wide range of Reynolds numbers all the way from attached flow cruise conditions throughout the flight envelope.

The main purpose of this report is to review/establish the baseline, or state-of-the-art, in aerodynamic flight prediction using computational fluid dynamics (CFD). One dictionary definition of state-of-the-art is: “the latest and most sophisticated or advanced stage of a technology, art, or science.” However, because the most sophisticated or advanced stage is sometimes debatable, the term is often used instead to refer to the latest most common accepted usage, and not necessarily the most advanced. It is this latter definition that primarily applies to this report. A second purpose of this report is to identify gaps and needs for future research.

Today, CFD methods provide predictions for a wide range of purposes during development of subsonic fixed wing aircraft. The range of purposes is so broad, however, that covering all of its facets to any meaningful depth would be impossible in a report of this nature. For example, in the “Decadal Survey of Civil Aeronautics: Foundation for the Future,”² many specific research and technology challenges in addition to aerodynamics were identified, including

aeroacoustics, propulsion and power, materials and structures, dynamics, navigation, control, and avionics. Even the top ten challenges identified in the area of aerodynamics and aeroacoustics alone represented a broad range of needs for which CFD plays an important role, including integrated system performance, separation control, transition and turbulence modeling, icing, and wake vortex prediction, detection, and mitigation.

Therefore, in light of the daunting breadth of research performed/needed in CFD in support of subsonic fixed wing aircraft, this report instead focuses on two specific areas: cruise through buffet onset in the transonic flow regime, and flow control for high lift at lower speeds. Both of these are key areas for which improvements in CFD capability and reliability are needed. It is believed that by demonstrating and discussing the state-of-the-art as well as gaps and needs in these two areas, an adequate overall impression of the status and needs of the broader CFD field for subsonic fixed wing aircraft as a whole will emerge. Specific issues such as turbulence modeling for RANS or subgrid scale modeling for LES are not addressed directly here, other than as they relate to the areas of cruise through buffet onset and flow control for high lift (the interested reader is referred to modeling surveys³⁻¹¹ for more specific information). Both areas will be covered by describing results and lessons learned from specific workshops or studies. Also, because it plays such a fundamental role in the process of assessment, uncertainty quantification (often referred to as verification and validation) will be described in some detail in the next section.

II. Uncertainty Quantification

This section covers the current status of uncertainty quantification, or being able to give an estimate for “how good” a given CFD simulation is. Often uncertainty is thought of as the ability to associate an error band with the solution. As will be seen, this capability is currently not well defined today. There is even confusion concerning the meanings of many of the terms used to describe it. (In fact, as discussed below, the word “uncertainty” itself is often assigned a specific meaning, but its broader use is already widespread.) This discussion is not limited to subsonic fixed wing aircraft, or even to aerodynamics. This broad field applies to computational simulations across a wide range of disciplines, and its issues and problems are common to each. However, we will only refer to its use for CFD throughout this discussion.

As the use of CFD as part of the vehicle design process has increased in recent years, uncertainty quantification for simulations has grown in importance. Unfortunately, this type of evaluation is not always straightforward, and there is significant ongoing research currently dedicated to developments in this area. This field is closely related to “Verification and Validation,” or V&V. As discussed in Roache,¹² uncertainty quantification often refers to V&V, but also allows for the inclusion of error estimation, banding, and factor of safety. For the purposes of this discussion, we will not draw any distinctions. According to the AIAA Guide for V&V,¹³ the term verification refers to the process of determining that a model implementation accurately represents the developer’s conceptual description of the model and the solution to the model. The term validation is the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model. In essence, then, the process of verifying a code is determining if the model is solved correctly (e.g., no bugs in the software), and the process of validating a code (usually for a specific range of problems) is determining if the model is a good one to use for the intended purposes. Note that because many CFD codes are so complex, verification and validation are usually an ongoing process.

The AIAA Guide also distinguishes between the terms “uncertainty” and “error.” The former is a potential deficiency due to lack of knowledge, whereas the latter is a recognizable deficiency *not* due to lack of knowledge. Examples of uncertainty might include not knowing the validity of a turbulence model or not knowing the precise shape or surface roughness of a wing. Examples of error includes both acknowledged sources (finite precision, insufficient spatial discretization, insufficient iterative convergence) and unacknowledged sources (such as use of incorrect input files or other mistakes). Uncertainty and error occur in *both* CFD and experiment, often making it difficult to distinguish the cause(s) of any disagreements between the two. A graphic illustrating these concepts, adapted from a figure in Luckring et al.,¹⁴ is shown in fig. 1.

Most CFD code users implicitly assume that the codes they are using have undergone verification assessment. This is probably only true in a very small percentage of cases, in part because the process can be difficult, particularly for boundary conditions and turbulence models. The method of manufactured solutions is a popular method for performing verification.¹⁵⁻¹⁸

Assuming that the CFD code being used has been verified, and trusting that unacknowledged sources of error are small, the goal is to quantify both the uncertainty and the acknowledged sources of error. The latter is generally the easiest to achieve in theory, because it relies solely on the numerical solutions and not on any comparison with experiment. For steady state problems, most acknowledged error comes from discretization error (insufficient grid

resolution or poor grid quality); it is usually analyzed through grid sensitivity studies. Unfortunately, in practice the discretization error can be difficult to quantify for complex problems. A proper grid sensitivity study can be difficult to perform, because: (1) quite often it is impossible to guarantee that the “asymptotic range” of grid convergence has been reached (which is necessary in order for methods such as Richardson extrapolation and the Grid Convergence Index to be meaningful¹²), and (2) it can be difficult to create a series of grids of the same or similar family. Much active research is being dedicated to quantifying errors associated with grid resolution, a few of which are referenced here.^{19–21} Other methods, such as use of adjoints²² and use of downscaling tests,²³ have been used to attempt to quantify these types of errors. Eventually, the best approach for simultaneously quantifying *and* reducing discretization error may be automatic grid adaptation based on some method like adjoints.^{24,25} But these methods are currently not robust enough for general use with high-Reynolds-number RANS.

Validation is typically performed by comparing CFD with a variety of theoretical or experimental databases. For real-world experiments, having a wide array of well-performed, well-documented, error-bound experiments for a given class of problem would be best, but this is typically not the case. And even with well-defined experiments, validation processes can be difficult to define.^{26,27} A significant fraction of the battle seems to be setting up the CFD to run the same problem (geometry and boundary conditions) as experiment. Often, these quantities are not precisely-enough defined to the degree needed by the CFD. This lack of knowledge typically inhibits the ability to assign the root cause of differences to modeling problems such as inadequate turbulence model. Fig. 2 shows some characteristics of four different validation phases, illustrating how increasingly complex experiments include more complex flow physics, but also yield less data and larger uncertainty.

Some useful computational techniques, such as the aforementioned adjoint methods (usually employed in the context of design^{28,29}) have been developed for determining sensitivities in the solution to small changes or uncertainty in physical properties or numerical parameters (such as model constants). Other methods include but are not limited to polynomial chaos,³⁰ sensitivity equations,^{31,32} and sampling methods.³³

Regarding uncertainty quantification, the bottom line of the current state of affairs is that it is not very well defined, performed, or understood in general. As a result, most workshops that attempt to quantify uncertainty or establish the current state of CFD for specific classes of physical flows end up being somewhat inconclusive. See, for example, Eca et al.³⁴ Simply put, it is difficult to sort out all the various sources of uncertainty and error. Luckring et al.¹⁴ provided an approach for moving toward uncertainty-based CFD. It involved both attempting to achieve statistical process control within the context of CFD studies, and managing the uncertainty process (from the decision maker and resource allocation perspective). The latter approach was represented by five uncertainty stages, reproduced in fig. 3. NASA has also developed a (currently interim) technical standard for models and simulations,³⁵ which defines requirements for the development of models, operation of the simulations, analysis of results, training, recommended practices, assessment of credibility, and reporting.

Although progress is being made in certain areas and for specific classes of flows, overall the current practices within the aerodynamics community in uncertainty quantification/V&V are inadequate. Grid convergence studies (particularly in 3-D) are often flawed because of improper construction and usage of grids.³⁶ Much of the problem is likely the fact that it is extremely difficult to build a proper sequence of grids for complex configurations. It is not even known how to define “proper sequence” for unstructured grids in general. More routine verification of CFD software is needed,³⁷ and better uncertainty quantification processes need to be established and accepted by the community as a whole before validation results can become less ambiguous.

III. Cruise Through Buffet Onset

In this section, we focus on transonic (in the range of $M = 0.75 - 0.87$) flow over aircraft configurations, with separation prediction and/or separation onset and progression prediction as the primary focus. The main emphasis here will be on results from a series of drag prediction workshops (DPW) sponsored by the AIAA Applied Aerodynamics Technical Committee, and on results from a joint NASA/Boeing computational study on modern civil transport aircraft.

A. Overview

There is a significant need for accurate and reliable CFD predictions of transonic flow over aircraft configurations in industry today. At and near cruise conditions, CFD tools and processes (including grid generation, etc.) have advanced to a state that industry has come to trust RANS predictions to provide consistent predictive results for the most part^a. But outside the cruise regime, and especially near the “edges” of the flight envelope, CFD is not currently able to

^aE. N. Tinoco, Boeing Commercial Airplanes, private communication, [cited 9/2007]

deliver to the same level of confidence. See fig. 4, which shows a notional graphic representing the flight envelope as a function of load factor and equivalent airspeed. In two regions representing low-speed and cruise attached-flow aerodynamics, CFD is considered accurate, but in other regions the accuracy is not known. This is not to say that current CFD is not usable in off-design regimes, such as those characterized by significant regions of flow separation, including buffet. Often, it can be successfully used either in predicting absolute levels or in predicting trends. But its record is “spotty;” success seems to depend on the configuration or case, so its reliability is considered dubious.

It is recognized that not only does RANS CFD need to be numerically accurate (sufficient grid density), with good physical models (turbulence modeling), but it also has to be set up to physically match the wind tunnel or flight test in terms of geometry and flow conditions as closely as possible. For clean cruise-type flowfields, achieving these three conditions is currently well within the capabilities of state-of-the-art CFD. But for off-design conditions, the demands appear to be greater. It may be simply a matter of time, building up many case studies and collective experiences, and allowing the CFD community to determine best practice guidelines for different genres of off-design conditions. It is also possible that for certain conditions the current state-of-the-art turbulence models are fully inadequate, or even that the inherent assumptions made for RANS no longer apply.

Two detailed collective studies which attempted to determine CFD best practices for particular configurations are highlighted in this section. Note that in the first – the Drag Prediction Workshops – the configurations were nominally for cruise, but they contained one or more significant regions of separated flow, so separation was generally an (unintended) issue. These two isolated examples given here were chosen because of the authors’ familiarity with them. Clearly, the literature contains many additional examples. One study in particular is worth mentioning because of its strong focus on attempting to improve and document CFD methods and processes: the Abrupt Wing Stall Program.^{38–40} Only a few of the relevant references are cited here. Part of the goal of this program was to use CFD to predict an abrupt wing stall phenomenon, resulting from massive wing separation, that occurred on the pre-production F/A-18E/F aircraft. Early CFD attempts failed to predict the phenomenon adequately, but subsequent improvements in turbulence modeling, geometric fidelity, grid density, and solution run/convergence parameters yielded good predictions. This experience teaches us that early failure of CFD does not necessarily imply inadequacy of the tool itself; often it takes a careful study and a significant expenditure of resources to determine what the best practices must entail to compute a particular class of flows. The best practices that work for clean, attached flow do not necessarily work for separated flows.

B. Drag Prediction Workshops

To date, the DPWs have consisted of a series of 3 workshops, held in conjunction with AIAA meetings. They have garnered wide participation from many CFD groups around the world. Although originally geared toward the prediction of drag (as the workshop title implies) on realistic aircraft geometries, the series has also focused on prediction of lift, moment, and surface pressures, while comparing CFD results both to wind tunnel experiments and to each other. Summary papers have been written by Levy et al.,⁴¹ Laflin et al.,⁴² and Vassberg et al.⁴³ One unique aspect of the DPW has been the use of subsequent statistical analysis to process the CFD results.^{44–46} There have also been many papers highlighting individual contributions, some of which are listed here.^{47–69}

The first workshop used the DLR-F4 wing-body geometry at $Re = 3$ million based on mean aerodynamic chord. The contributed CFD results were rather disappointing in terms of consistency: there was more than a 270 drag count spread in the fixed- C_L data, with a confidence interval of more than ± 50 drag counts. This is significantly higher than the oft-quoted industry-desired accuracy of 1 drag count. One surprising result was a consistent overprediction in lift among almost all participants. Typical results⁵⁰ are given in fig. 5 (shown here for three different turbulence models on a 1-to-1 grid). The reason for the lift overprediction was not known. However, as discussed in Levy et al.,⁴¹ even though the tunnel data were corrected to a free air condition, the correction process introduces some error. Also, at this Reynolds number the flow is not immediately turbulent at the leading edge, whereas most CFD simulations were run “fully turbulent” (for which the transition location is model and code dependent). Poor grids supplied for the use of the workshop participants (not sufficiently fine, and poor orthogonality and smoothness characteristics) were a potential source of problems in many of the CFD simulations.

The second workshop used the DLR-F6 wing-body and wing-body-nacelle-pylon geometries at $Re = 3$ million. The latter configuration is shown in fig. 6. One specific objective was the prediction of incremental drag due to the addition of the nacelle-pylon. Another was to assess the effect of specifying transition location on the wing, nacelle, and fuselage. A third was to conduct a grid convergence study. Overall, although the code-to-code scatter in drag was significantly reduced compared to DPW-I, it was still an order of magnitude larger than desired by the aircraft industry.

DPW-II asked for more information from the participants than DPW-I, including surface pressure coefficients. One interesting result to come out of DPW-II was the fact that generally if a code ran to match C_L , then its surface

pressures would not agree with experiment; whereas if a code ran at a specified angle-of-attack, its surface pressures would agree with experiment but C_L would be too high. This inconsistency with experiment is the same problem seen for DPW-I (overprediction of lift). Typical surface pressure coefficient results are shown in fig. 7. Specifying transition location did not resolve the discrepancy.

The supplied grids again were problematic for the second workshop. Although sequences of successively finer grids were available, many of these were inconsistently refined and contained many areas of poor quality. An example from Rumsey et al.⁵⁵ showing the effect of the grid can be seen in figs. 8 and 9. The first figure shows separated streamlines on the underside of the wing inboard of the nacelle resulting from use of an overset grid, while the second shows no separation on a 1-to-1 grid. The same code and turbulence model were employed in both cases. The overset sequence of grids were significantly finer in this region (even the coarsest overset grid had greater grid density in some regions than the finest 1-to-1 grid), so it is likely that its results were numerically more accurate.

The second workshop also began to pay attention to the fact that there were several regions of separation for the configurations in the experiment: inboard of the pylon, along much of the upper surface trailing edge, and at the wing-root juncture. As discussed above, not all computed results showed separation inboard of the nacelle. Also, not all computed results showed upper surface trailing edge separation (generally results on unstructured grids did and results on structured grids did not). But most computed results showed a wing-root separation bubble. The computed size of the wing-root juncture bubble (both in the second and third workshops) was noted to be significantly affected by certain numerical parameters, such as thin-layer approximation vs. full Navier-Stokes. Example results are shown in fig. 10. The grid has also been shown to have an influence (with finer grids yielding larger bubble size).

Rumsey et al.⁵⁵ attempted to quantify the magnitude of various effects, including grid type (1-to-1 vs. overset), code (CFL3D vs. OVERFLOW), turbulence model (SA, SST, or EASM), transition (specified or free), and viscous model (thin vs. full). Results are shown in fig. 11 in terms of drag count for the wing-body and the wing-body-nacelle-pylon configurations. Although all of these variations are probably not additive, it is not surprising that the DPWs showed variations among participants of 25 drag counts or more, given these individual levels.

The third workshop worked to improve on the previous workshops in several ways. First, in an attempt to see if separation was the primary cause of grid convergence problems, the DLR-F6 configuration was modified with a side-of-body fairing (FX2B), which was designed to eliminate the wing-root bubble. Second, a new set of wing-alone cases was included, in an attempt to provide a simpler set of configurations. Third, the cases were all blind, in that no experimental data were provided or available at the time (Re for these cases was 5 million based on mean aerodynamic chord). And fourth, greater effort was put into the construction of the grids.

Unfortunately, there were still some problems identified for the grids. High grid quality and generating a consistent set of grids for the purpose of grid-convergence studies remains very difficult to achieve for 3-D configurations. In spite of this difficulty, many positive things came out of DPW-III. For one thing, there exists a core set of CFD methods that consistently agree with each other (both structured and unstructured solvers). Although large variations in predicted separation bubble sizes were seen among all the participants, the existence of the separation itself did not appear to adversely affect the grid-convergence trend of many of the computed results. The DLR-F6 and FX2B configurations demonstrated a monotonic decrease in variation with increasing grid resolution for the “core solutions” (the collective of fully nested solutions with outliers removed), as shown in fig. 12. Although not shown, the wing-alone cases demonstrated a nearly grid-independent variation for total drag when the lift component of drag was removed: $C_{D,idealized} = C_D - C_L^2/(\pi AR)$. The level of this variation was similar to that on the finest grids for DLR-F6 and FX2B.

In summary, the complete series of DPWs has been extremely useful in helping to gage the level of uncertainty present in state-of-the art CFD for these types of aircraft configurations. Use of statistical analysis can assist in identifying outliers, which can then be ignored when trying to establish quantitative uncertainty assessments from the collective. The quality of the sequence of grids created for CFD simulations is paramount when performing studies of this kind; apparently today’s grid generation tools or user skills are somewhat deficient, if the DPWs are an indication of the current state of the typical capability in this area.

C. NASA/Boeing Buffet Onset Team Results

In the early 2000’s, a joint NASA/Boeing team was formed to explore CFD issues associated with computing separated flow near buffet onset for a modern civil transport aircraft. Results were published in Rumsey et al.⁷⁰ Subsequently, an additional similar study was conducted for a different civil transport aircraft.⁷¹ These studies were conducted in large part to quantitatively assess the sensitivities due to grid, turbulence model, and code for aircraft buffet onset flow conditions. The main motivation came from earlier unpublished CFD studies in industry that seriously underpredicted buffet onset lift levels compared to flight test results. Blame was often attributed to turbulence modeling inadequacies.

The NASA/Boeing team study attempted to quantify not only the uncertainties due to turbulence model, but also to identify (and assess if possible) all of the other possible sources of error that arise when attempting to compare CFD simulations with either wind tunnel experiments or flight test results.

In terms of CFD, assuming that use of the RANS equations is appropriate for a particular case, three general categories of uncertainty were identified: numerical errors, modeling errors, and geometric fidelity. Relating these back to the terms defined in the AIAA Guide to V&V,¹³ they fall under the categories of acknowledged error, CFD uncertainty, and experimental uncertainty, respectively. The latter category of geometric fidelity is very broad, and can be influenced by the inability to adequately define the exact shape of a vehicle under load (approximations and assumptions are often made). Furthermore, Rumsey et al.⁷⁰ discussed some approximations made when extracting or generating flight test data (often obtained from a flight simulator). For example, in a process termed “rigidification,” the flight force curve is modified using linear scaling techniques to approximate what it would be if aeroelastic deflections were removed. Also, for off-design conditions, flight pressure data is often obtained through unsteady maneuvers, which may yield inconsistent results between the inside and outside wings and thus present difficulties in interpretation when compared to steady wind tunnel or CFD results.

An important aspect of these Buffet Onset Team studies was to establish and apply a consistent set of best practices for CFD. These guidelines, which arose from a group consensus of the experienced CFD practitioners from NASA and Boeing, are repeated here:

- Approximate the physical model (geometry) as closely as possible in the CFD grid. If walls, brackets, supports, etc. are ignored, recognize the fact and have a good idea of the effects that might be missed.
- Be aware of the aeroelastic deformations of the model or flight vehicle, and account for these in the CFD grid or be aware of their effects.
- Use a sufficient grid resolution and perform a grid sensitivity study to estimate discretization errors.
- Use correct boundary conditions. If the case is a wind tunnel comparison, investigate the effect of wind tunnel walls. If it is a free-air computation, ensure that the far field boundary condition is applied sufficiently far away.
- Maintain grid smoothness (quality) in structured grids as much as possible: maintain reasonable orthogonality, avoid too-large stretching factors and too-rapid turnings or twistings of grid face normal directions.
- Do not leave a gap in the wake cut behind a blunt trailing edge, particularly for attached flows.
- Cluster grid lines near the wing trailing edge in the streamwise direction to be less than or equal to approximately 0.2% of the local chord. Cluster grid lines near wing leading edge in the wrap-around direction to be less than or equal to approximately 0.1% of the local chord.
- Set the minimum spacing at walls to be small enough such that the y^+ levels are near 1 or less for turbulent flows.
- When computing turbulent flows, always verify the resulting computed transition location by looking at eddy viscosity contours or some other appropriate measure. Running “fully turbulent” does *not* guarantee transition to turbulence at the leading edge; the actual trip location is a function of the turbulence model and the Reynolds number.
- Spread grid lines in the wing wake (but not too rapidly); approximately follow the trailing edge bisector angle to try to align the grid lines with the local flow direction.
- Using more than one CFD code for a given case can help lend confidence to the validity of the results and also give an idea of the magnitude of differences due to different numerical treatments; however, be aware of differences in the *turbulence model equations* employed (i.e., be aware of turbulence model versions and implementation differences used by different codes).
- Determine the pedigree of experimental and flight data. Corrections are often applied to the data that limit its usefulness in direct comparison to CFD results. Do not trust experimental data blindly.

The study also summarized a list of general observations and lessons learned from previous industry studies, which are repeated here. Some of these are related to the best practice list generated by the team.

- 2-D airfoil testing is not really 2-D, even when there is sidewall suction: For example, CFD misses trends and shock location for a high-wing transport type airfoil. CFD results improve if run in 3-D and sidewall suction present in the experiment is modeled.
- When modeling thick trailing edge wings, it is better to “close the wake” with the grid (many CFD users opt to leave a gap in the wake – this improves convergence, but also adversely affects shock location and misrepresents possible real unsteady physics).
- Upwind differencing is generally recommended over central differencing with scalar dissipation. Central differencing tends to smear shocks, underpredict suction peaks, and overshoot total pressure.
- The Baldwin-Barth turbulence model exhibits problems (kinks) near the edge of boundary layers that get worse with grid refinement.
- There are inherent difficulties comparing CFD with low Reynolds number wind tunnel data because of uncertainty of transition location on the tunnel model.
- CFD is not reliable for estimating flight Reynolds number aileron effectiveness characteristics when flow separation is involved.
- For airframe/engine integration, CFD enables the design of interference-drag-free installations, and gives excellent insight into Reynolds number scaling effects for attached flows; but there is less confidence for predicting separated flows.
- Actual digitized (as-built) geometry can be different from a design shape; using CFD with the digitized geometry improves comparison with experiment.
- Including the sting in a CFD simulation can be important for some wing/body configurations: it can affect shock location on the wing.

Established CFD variations for the two studies are shown in figs. 13 and 14. The first figure shows the maximum variation in lift, drag, and pitching moment for the four categories of grid size (error compared to extrapolated results on grid of infinite density), code or differencing method, tunnel model aeroelastics, and turbulence model. The turbulence model (effect between 4 state-of-the-art models SA, SST, BB, and EASM) yields the largest variation, with code/differencing typically second largest. If all four categories were additive (worst case scenario), then the combined approximate maximum variation of computed forces and moments due to individual differences in code, spatial differencing method, and turbulence model were: 6% in lift, 7% in drag, and 16% in pitching moment.

For the second configuration shown in fig. 14, the “G” no longer refers to effect of overall grid density, but rather to the specific effect of additional trailing edge grid resolution (through the use of a “cap grid”), and aeroelastic effects were not evaluated. In this case, the higher the angle-of-attack, the larger the variation (in general). Overall, similar trends were seen for the two configurations. Turbulence model again gave the largest of the individual effects. If all three categories were additive (worst case scenario), then the combined approximate maximum variation of computed forces and moments due to individual differences in code, spatial differencing method, and turbulence model were: 4% in lift, 3% in drag, and 31% in pitching moment. The reason for the significantly larger percentage variation in the moment compared to fig. 13 is due to the fact that the absolute moment values for the current configuration were approximately half those of the previous configuration. The absolute variations in moment levels were about the same.

The main results from these studies comparing CFD results with flight test data are given in figs. 15 and 16. Details concerning various grids and methods can be found in the respective references. The bottom line of both of the investigations is that the problem of CFD being unable to reach flight buffet levels did *not* occur for the current two aircraft configurations. Instead, the predicted lift curve from CFD tracked the trend from flight data well through buffet all the way to near maximum lift. If anything, the relatively small error band due to different codes and turbulence models for a separated flow buffet-onset case, combined with reasonably good agreement with experiment, suggests some validity of today’s state-of-the-art CFD tools for aerodynamic flows outside of the cruise envelope.

The reasons for the success of CFD in this case as opposed to the failure of CFD in earlier unpublished studies in industry are not known. It is possible that the flow fields of the configurations in the earlier studies were more sensitive to small perturbations near buffet onset than the flow field of the current configurations. Part of the difference may also be due to the use of inappropriate flight data: the use of simulator-derived flight data as well as the use of a rigidification process appear to be questionable for comparing to wind tunnel or CFD results. Finally, many of the

geometric simplifications and omissions necessary when performing CFD analysis of a complex flight vehicle are a potential source of error.

State-of-the-art CFD is by no means fully adequate for predicting separated flows and buffet onset for aircraft configurations. There are significant variations among modern RANS turbulence models, and no doubt none of them is able to faithfully represent (in the mean) the complex physics of separated flows, which is inherently unsteady. However, the studies described here suggest that current RANS methods can be successful to a certain degree in modeling trends at off-design conditions. Unfortunately, there are many potential sources for disagreement when trying to compare CFD results with wind tunnel tests or flight data, and it is very difficult to account for them all. As a result, it may be impossible to ensure that a CFD simulation is close enough to the “real thing” in terms of geometry and boundary conditions. Through careful CFD studies that make use of best practices, many of the uncertainties can be reduced, but at the present time it seems that it is only for relatively simple configurations and testing (so-called “unit problems”) that one can truly be confident of running the same problem as experiment, and thus draw firm conclusions regarding specific inadequacies in turbulence models.

D. Gaps and Needs

If state-of-the-art CFD is not fully adequate for subsonic fixed wing aircraft operating in cruise through buffet onset conditions, then precisely how does it fall short? We have seen that, in isolated cases, it can perform reasonably well even for separated flow conditions. And industry tells us that they have come to trust CFD tools and processes for certain flow regimes. Looking at contributions to the DPW series of workshops, it appears that the aeronautics community has by-and-large moved to general acceptance and use of the Spalart-Allmaras⁷² or Menter $k-\omega$ SST⁷³ turbulence models. Does this mean these models are known to be adequate for aerodynamic flows in general?

The answer to this last question is clearly “no.” Using unit problems, which largely eliminate many of the uncertainties associated with experiments (so that CFD can “run the right problem”), it has been possible to better isolate the models as the source of differences between CFD and experiment. An example of a fairly definitive failing of the Spalart-Allmaras and Menter $k-\omega$ SST (as well as other) RANS turbulence models will be described later.

It appears that the difficulty associated with the two collective transonic studies described above is that the various sources of uncertainty have been extremely difficult to separate from each other and from acknowledged and unacknowledged error sources. As a result, it cannot be said *with certainty* that the current state-of-the-art turbulence models are necessarily faulty with respect to the given studies. In other words, it is not known whether any definitive failings that have been identified through unit problems necessarily have a significant effect for some of the characteristics of interest for full-aircraft configurations in cruise-through-buffet conditions.

A need that *has* been identified is improved processes for helping to reduce or eliminate error sources, allowing systematic evaluation of uncertainty. Production aircraft companies seem to have achieved some level of success in this area, by strictly controlling their CFD process all the way from grid generation to CFD flow solution and postprocessing. It is not known how to attain this same level of control within the very broad aerodynamics community as a whole. Publication and dissemination of guides or “lessons learned” documents can certainly be of some use. Eventually, specific tools such as automatic grid adaptation – when it becomes widely-used – will be key in helping to achieve the goal of quantifying uncertainty. In the nearer-term, improved grid-generation processes and practices, particularly for creating appropriately-refined sequences of successively finer grids, would help reduce much of the grid-induced error and make analysis easier.

Experimental needs (for aiding CFD validation) appear to be mostly associated with reducing ambiguities in geometry and boundary conditions for complex configurations. The more confident one is that these aspects are properly accounted for in the CFD simulation, the easier it is to isolate modeling issues.

Finally, it is worth noting that workshops and validation exercises such as the DPW series and the NASA/Boeing Buffet Onset Team effort are extraordinarily valuable. Even though results are sometimes not as clear-cut as hoped, many benefits always emerge. A segment of the aeronautics research community (often made up of both CFD and experimental experts) comes together and focuses on a fairly specific issue. This focus is very useful, because it helps to isolate systemic problems and develop a shared understanding of where both the CFD and experimental communities need to concentrate their efforts and resources. Unfortunately, some organized exercises occur primarily because of the passion and dedication of its volunteer participants, and not because of institutional support (time or money) from the organizations to which they belong. Perhaps a definitive need here is an increased focus – or a more deliberate, active, well-funded support – of coordinated verification and validation workshops in general, on the part of management and funding sources.

IV. Flow Control for High Lift

In this section, we focus on the use of flow control, employed primarily for lift augmentation and separation control. Most of the applications of flow control are for low-speed flows. The main emphasis here will be on two recent flow-control workshops, from which we glean information on the state-of-the-art CFD capabilities in this area.

A. Overview

Generically speaking, flow control is the method for altering a natural flow to a more desired state. There are many methods for flow control, including both passive and active control techniques. Gad-el-hak⁷⁴ provides a summary of many of these in his book.

Flow control has many potential applications related to high lift. The use of pulsed flow control has the potential for reducing wing area, reducing part count, lowering weight, and reducing runway take-off and landing requirements.⁷⁵ Although steady-state blowing and suction can also be effective methods for flow control, complex ducting systems may be required. Synthetic (or zero-net-mass) jets can eliminate much of the complexity. Furthermore, it has been found that unsteady flow control can save 90-99% of the momentum required to obtain similar gains in performance using steady blowing.⁷⁶

In the Air Force, flow control technologies are being aggressively explored for both internal and external applications, with a significant portion of the effort focused on mechanical, pneumatic, piezoelectric, and electromechanical systems.⁷⁷ Goals include maintaining attached flow over wings and control surfaces, increasing maneuverability in parts of the flight envelope, and managing engine face flow distortions. Flow control also finds many applications for undersea vehicles, both in terms of reducing acoustic signature⁷⁸ and increasing maneuverability.^{79,80}

Clearly, vehicle designers would like to be able to rely on CFD as an aid to developing flow control systems. The use of CFD for flow control applications has been steadily increasing over the last decade. In particular, along with increasing computer power there has been increasing use of CFD for simulating unsteady problems. The very expensive large-eddy simulation (LES) and direct numerical simulation (DNS) methods have also made their way into this arena. However, even for the simpler steady-state flow control problems, CFD has not yet demonstrated itself to be reliable as a predictive method. Below, we look at two specific recent flow control workshops, and establish state-of-the-art snapshots of the current capabilities of CFD for these applications. The applications cover: (1) steady blowing over a Coanda surface for augmenting lift and increasing maneuverability, and (2) steady and synthetic jet flows primarily geared toward separation control. Obviously, there are many other flow-control-type of applications not included in this assessment. However, it is often difficult to assess individual isolated results from the literature; whereas focused workshops such as those described here provide a common point of reference, making assessments easier.

Note that there have also been many efforts focused on computing high lift flows *without* flow control. See Rumsey and Ying.⁸¹ Conclusions from this survey were that CFD often disagreed with experiment at maximum lift conditions, with the main reasons primarily being numerical errors and lack of geometric or modeling fidelity (for example, using 2-D simulations when experiments near stall are likely no longer two-dimensional). Modern turbulence models – such as SA, SST, EASM, and others – were found to perform better for separated flows, and many $k-\varepsilon$ models were determined to be poor choices for adverse pressure gradient wall-bounded flows. Transition locations in CFD could play an important role. Conclusions were similar for a European assessment.⁸²

B. Circulation Control Workshop

With circulation control, a tangential wall jet is used primarily for the purpose of enhancing lift over an aerodynamic surface. A wall jet emanating from a plenum inside an airfoil or wing “sticks” to the rounded trailing edge surface due to the Coanda effect, causing delayed flow separation and thus increasing circulation and producing higher lift. This type of active flow control can provide substantial benefits for real-world applications, as demonstrated for the V-22 tiltrotor aircraft configuration.⁸³ Organizers of the 2004 Circulation Control Workshop⁸⁴ held in Hampton Virginia asked CFD participants to compute flow over the NCCR 1510-7607N airfoil, with blowing over its elliptical (Coanda) trailing edge. This circulation control (CC) experiment was conducted in 1977 by Abramson.⁸⁵ Some participants computed flow over a similar configuration, the 103RE(103XW) airfoil, tested by Abramson and Rogers.⁸⁶ A graphic showing the trailing edge of the airfoil configuration is shown in fig. 17. For the blown conditions, all participants at the workshop used Reynolds-averaged Navier-Stokes (RANS) and a variety of turbulence models, including one-equation linear, two-equation nonlinear, and full Reynolds stress.

One of the main conclusions to come out of the workshop was the inconsistency in the CFD results for capturing the Coanda jet flow physics (especially separation). In particular, many RANS computations could – for some blowing conditions – obtain unphysical results, where the jet wrapped around the lower airfoil surface too far. An example is shown in fig. 18. Furthermore, even when different computer codes used (ostensibly) the same turbulence model, significant differences were seen between the reported results.

Many recent papers have been published on CC airfoil flows.^{79,80,87–97} The bottom line seems to be that (a) for RANS, some turbulence models (such as Menter $k-\omega$ SST,⁷³ SARC,⁹⁸ $k-\zeta$,⁹⁹ explicit algebraic stress,¹⁰⁰ and full Reynolds stress models¹⁰¹) *can* do well for certain conditions, but it depends on the case and there is a tendency for the solutions to degrade compared with experiment as the blowing increases (see, for example, fig. 19); and (b) these CC flows tend to be very sensitive to numerical parameters. For example, Swanson and co-workers demonstrated high sensitivity in the SST model to how the production term was computed⁹⁴ as well as to grid density.⁹⁵ Transition location of the jet could also be important^b. Other types of computations, including DES⁸⁸ and LES¹⁰² have been rather isolated and too preliminary to draw any firm conclusions about their potential for this class of CC flows. For example, in the LES computation, the jet separated too early. This deficiency was believed to be due to insufficient perturbations in the jet outer layer, causing too-weak streamwise instabilities in the jet. It is possible that improved LES boundary conditions for the jet could improve the results.

Boundary conditions have also been seen to play an extremely crucial role in RANS flow control validation studies conducted since the time of the CC workshop. One important issue concerns the jet momentum coefficient. In CC and other flow control experiments, the jet momentum coefficient is often the only information given regarding the jet boundary condition. It is defined as: $C_\mu = (\dot{m}u_j)/(\frac{1}{2}\rho_\infty U_\infty^2 L)$, where the jet velocity u_j is typically obtained from conditions inside the plenum combined with isentropic flow relations. Unfortunately, this methodology introduces a degree of uncertainty into the CFD simulations. It is far more useful to the CFD community to have measured values of the flowfield *at the jet exit* in the measurement plane, including velocity (and turbulence properties if possible). Furthermore, for nominally two-dimensional configurations, it is important to establish and quantify how far the experimental flowfield deviates from two-dimensionality. For example, often the flow rate is only measured as it enters the wind tunnel test section, but it is also important to determine how uniformly (or nonuniformly) it exits the plenum slot. Any 2-D CFD simulation needs to accurately mimic the local jet boundary conditions in the measurement plane.

It also became evident at the workshop that more CC experiments are needed for validation. The data used for this workshop were from experiments conducted nearly 30 years ago. It was further recognized that this type of experiment is difficult to perform, particularly when trying to maintain two-dimensionality as blowing increases. Often at the higher blowing rates, sidewall vortical structures can significantly affect the flowfield, inducing downwash and affecting the effective angle-of-attack of the airfoil.

C. CFDVAL Workshop

The Langley Research Center Workshop on CFD Validation of Synthetic Jets and Turbulent Separation Control (also known as CFDVAL2004) was held in Williamsburg Virginia in 2004.¹⁰³ This workshop brought together both experimentalists and CFD experts, and three flow control experiments were designed and performed specifically for the workshop. Furthermore, the experimental data and workshop results were posted to a public website^c, which has subsequently encouraged a great deal of additional research and published papers on the test cases. To date, there have been nearly 40 CFD papers published related to the CFDVAL2004 workshop. A partial list – of only the journal articles published to date – is given here.^{104–114}

The three workshop cases were chosen to represent different aspects of flow control physics: nominally 2-D synthetic jet into quiescent air, 3-D circular synthetic jet into turbulent boundary-layer crossflow, and nominally 2-D flow-control (both steady suction and oscillatory zero-net-mass-flow) for separation control on a simple wall-mounted aerodynamic shape. It is important to note that experiments are difficult to perform for these types of unsteady flowfields. In the experiments here, an effort was made to take duplicate measurements using different techniques; this duplication highlighted the uncertainties inherent in the measurements.^{115–119} A summary of the workshop results can be found in Rumsey et al.¹⁰³ Here we briefly recap the main conclusions, then summarize results of additional CFD research that has occurred subsequent to the workshop.

Case 1, 2-D synthetic jet into quiescent air, was a difficult experiment to simulate. The flowfield was probably partially laminar or transitional, so it was unclear how best to simulate it. Workshop participants used RANS, laminar

^bR. C. Swanson, Jr., NASA Langley Research Center, private communication, [cited 9/2007]

^c<http://cfdval2004.larc.nasa.gov> [cited 9/2007].

Navier-Stokes (N-S), blended RANS-LES, LES, and a reduced order model. End effects probably caused significant three-dimensionality far away from where the jet emanated from the wall, but most participants computed the flow in 2-D. The piezoelectric driver was a difficult device to simulate, so most computations made approximations inside the plenum or simply applied jet boundary conditions directly on the wall from which the jet emanated. As a result, the CFD simulations did not even start off with the same conditions as each other or as the experiment at the jet exit: deviations from periodicity were for the most part not simulated. For RANS, it was therefore difficult to judge the capabilities of the turbulence models. However, Carpy and Manceau,¹⁰⁶ who used extracted PIV data near the slot exit as surface boundary conditions, later noted that full Reynolds stress models offered improvements over linear models when representing turbulence dynamics for this case, because they can capture the presence of a region of negative production that occurs during the deceleration phase. They also brought up the point that when applying surface boundary conditions for the jet (i.e., not solving the flow in the cavity), prescribing an inlet value for ε can be problematic.

It is important to note that additional measurements taken after the workshop, given in Yao et al.,¹¹⁵ were at somewhat different conditions than those used for the workshop case. Some of the larger discrepancies exhibited by different measurement techniques in the original data were mitigated in the later experiment.

Typical results from the workshop are shown for mean jet centerline velocity in fig. 20. This figure exhibits the wide range of results obtained by the participants. It is an indication of the large uncertainty inherent in CFD for a simple synthetic jet application. However, it should be noted that the grids used by some participants were coarser than those used by others, so some of the variation may have been caused by numerical under-resolution. Also, as mentioned above, jet boundary conditions were not consistent between the participants.

Subsequent investigations performed with 2-D RANS by Vatsa and Carpenter,¹²⁰ Vatsa and Turkel,¹⁰⁵ and Park et al.¹²¹ indicated generally improved comparisons with experiment. The primary cause for the improvement was likely the increased attention given to achieving similar velocity profiles to experiment at the jet exit. All of these computations modeled a simplified plenum with periodic transpiration applied on the bottom wall, but the imposed velocity was curve-fitted to better replicate the experiment. In particular, Vatsa and co-workers obtained nearly the precise temporal variation of the experimental signal by curve-fitting the measured velocities at the slot exit with a fast Fourier transform to reflect the proper mode shapes and to ensure zero net mass. As a result, they were able to obtain mean flowfield results in very close agreement with experiment. An example using the Spalart-Allmaras turbulence model⁷² is seen in a plot of time-averaged vertical velocity along the jet centerline in fig. 21.

Simulations using the 3-D incompressible (laminar) N-S equations have been performed by Kotapati and Mittal¹²² and Kotapati et al.¹¹⁴ They modeled an approximate plenum shape with periodic spanwise boundary conditions, and claimed that their computations captured the transitional nature of the flowfield. Xia and Qin¹²³ performed 2-D laminar and 3-D DES simulations with periodic spanwise boundary conditions. Unlike most others, they attempted to model the geometry of the piezoelectric driver (in a 2-D sense) with greater fidelity and included a wave-like function on the velocity profile applied at the plenum side-wall where the driver was located in the experiment. All of these 3-D simulations indicated that 3-D N-S (essentially an under-resolved DNS) and blended RANS-LES methods are also capable of simulating this type of flowfield. Unlike RANS, these methods hold the promise of more accurately predicting turbulence effects, through directly resolving the large eddy structures. It is not known whether accurate prediction of these turbulence effects for this type of flowfield is required or not. In any case, when comparing against experiment, faithfully mimicking the boundary conditions at the jet exit is certainly of primary importance for all methods: RANS, LES, and direct simulations alike.

For Case 2, a 3-D circular synthetic jet issued into a turbulent crossflow boundary layer. Most workshop participants used RANS, and one used LES. The experiment exhibited a large cross-flow velocity of unknown origin at the jet orifice exit, which was not modeled in any of the CFD simulations. Qualitative agreement with experiment was reasonably good, but quantitative comparisons showed significant variations. Different turbulence models were found to have less of an impact than different grids, codes, or other solution variants. Somewhat unexpectedly, LES and RANS solutions on similar-sized grids yielded very similar results in mean-flow quantities. However, as described in Dandois et al.,¹⁰⁸ LES gave better turbulent stress predictions.

Other works appearing subsequent to the workshop¹²⁴⁻¹²⁸ were mostly RANS (although Cui and Agarwal¹²⁶ also tried DES), and all used similar boundary condition methodologies (periodic vertical velocity imposed on the bottom wall of the plenum). Similar to the workshop, results among these papers seemed to vary widely. However, the RANS methodology was certainly capable of obtaining very good results for certain averaged quantities compared with experiment, as demonstrated in fig. 22. This figure shows the boundary layer perturbed in its phase-average by the passage of the synthetic jet structure. It indicates good agreement with experiment and relative agreement between three different turbulence models and two different grid sizes. It also illustrates the degree of uncertainty inherent

in the experiment; there were fairly large differences between results obtained using LDV and PIV at this particular location and phase.

Rumsey et al.¹²⁸ computed a somewhat different circular jet into crossflow¹²⁹ in addition to this one, and compared the two. Numerical effects were explored, such as the effect of grid size, time step, number of subiterations, symmetry vs. full plane, and effect of imposing jet boundary conditions on the floor. For the latter, it was found that use of a top-hat sinusoidal boundary condition at the orifice exit plane was an oversimplification that failed to capture the complex nature of the flowfield near the orifice. It was recommended to include at least some portion of the orifice in the computation. This conclusion is in agreement with earlier work of Rizzetta et al.,¹³⁰ who found that accounting for the internal actuator geometry in a synthetic jet computation was important because it affected the jet profiles at the exit.

Case 3 was flow over a nominally two-dimensional wall-mounted hump, inspired by earlier experiments of Seifert and Pack.¹³¹ Reynolds number was near 1 million based on chord. On this model, the flow (with no control) separates near 65% chord, and reattaches downstream past the end of the hump. Both steady suction control or oscillatory synthetic jet control applied near the separation point can lessen the size of the separation bubble. After the CFDVAL2004 workshop, the experimental data from case 3 were included as part of the ERCOFTAC on-line database (Classic Collection)^d, and were also included as test cases in two subsequent workshops: the 11th ERCOFTAC/IAHR Workshop on Refined Turbulence Modelling in Goteborg, Sweden, 2005, and the 12th ERCOFTAC/IAHR Workshop on Refined Turbulence Modelling in Berlin, Germany, 2006. Although not discussed here, results from these later workshops were consistent with the discussion to follow.

At the CFDVAL2004 workshop, 13 contributors ran 56 separate computations for Case 3. Methods were mostly RANS, but there were also blended RANS-LES results and one under-resolved DNS result. Two important conclusions were made at the time regarding this case. First, it was found that the side plates used in the experiment caused blockage that need to be accounted for in any CFD simulation, in order to obtain reasonable wall pressures over the attached portion of the hump. Second, nearly all models and methods at the workshop consistently predicted reattachment location to be too far downstream. This can be seen in the plot of reattachment location for the steady suction case in fig. 23. Here, DNS was an under-resolved direct simulation, LNS was a particular blended RANS-LES model,¹³² EASM-FSM was a blend of EASM and LES,¹³³ and v2f was a three-equation model.¹³⁴ Even the under-resolved DNS predicted a separation bubble that was too long. RANS grid refinement studies and use of methods with higher order spatial accuracy did not help. Inside the bubble itself, most RANS computations predicted velocity profiles in reasonably good agreement with experiment, but severely underpredicted turbulent shear stress in magnitude. Typical results (on two different grid sizes) are shown in fig. 24. DES results at the workshop¹¹⁰ (although not shown in fig. 23 because reattachment point was not reported at the time) also showed reattachment too far downstream for the suction case. However, results were better for the no-flow-control case, and it was surmised that DES may need to seed upstream eddy content into the boundary layer for shallow separations. Sarik et al.¹¹² also noted problems with DES for the shallow separation suction case, and said that this was possibly due to sensitivity of the method to grid design and the fact that the boundary layer upstream of separation was thinner. Streamlines using RANS with three different turbulence models are shown in fig. 25, demonstrating the typical overprediction of bubble length seen by almost all participants, for both steady suction and oscillatory control. Although not shown, RANS methods also overpredicted the bubble length when there was no flow control.

It should be noted that the RANS results for Case 3 were consistent with previous workshops using different massively-separated configurations. For example, at the 10th ERCOFTAC/IAHR/QNET-CFD Workshop on Refined Turbulence Modelling in Poitiers, France, 2002, two of the three test cases featured massive separation. RANS results were disappointing, and for a 2-D hill configuration (similar in nature to the hump case, but without any flow control) the RANS methods generally predicted too-long separation extent of the bubble, when separation location was predicted correctly. Documentation for a particular RANS submission to this workshop is given in Rumsey.¹³⁵ It should also be noted that these hump results are *not* consistent with those of a backstep, where the flow separates due to a sudden step, then reattaches downstream. Many turbulence models (only a few of which are referenced here) have been shown to do reasonably well in terms of predicting the turbulent shear stress and reattachment location in the separated region of a backstep.^{100, 136–139} Although the backstep flow also features a reattaching separation bubble, it differs compared to the hump and hill cases in that the latter two separate from a smooth surface due to adverse pressure gradient, whereas the backstep separates solely due to its geometry.

Since the time of the CFDVAL workshop, aside from a greater emphasis on the oscillatory (synthetic jet) case, no definitive progress has been made for RANS models in the sense that new results^{140–143} have been mostly consistent with results from the workshop. However, Rumsey and Greenblatt¹⁴⁴ conducted a RANS study to discern whether

^d<http://cfm.mace.manchester.ac.uk/ercoftac/> [cited 9/2007].

trends could be predicted, even though absolute levels could not. For steady suction, CFD appeared capable of predicting the trends due to Reynolds number and suction strength, and this was clearly evident by comparing bubble length and form-drag changes. Trends in bubble length and pressure drag coefficient relative to baseline (no control), due to varying strength of suction (C_μ), are shown in fig. 26. Three different turbulence models are shown to be in reasonably good agreement with experiment relative to its baseline, and computed bubble length trend relative to baseline is of correct magnitude but slightly shallower slope than experiment. For oscillatory control, the CFD indicated increasing effectiveness in the mean with increasing Re , but the effect was not nearly as pronounced as with steady suction. This difference was consistent with experimental observations. Overall, however, CFD did not mimic the mean effect of increasing oscillatory momentum coefficient very well, and although some effects due to changes in control frequency were captured qualitatively by CFD in the phase-averaged results, trends in the mean were missed.

Some work has been done in the area of blended RANS-LES, LES, and (under-resolved) DNS. For the latter, Postl and Fasel¹¹¹ doubled the spanwise domain extent (from $\Delta z/c = 0.071$ used at the CFDVAL2004 workshop to $\Delta z/c = 0.142$) and saw an improvement in their results, including reattachment length, compared to experiment. Furthermore, unlike RANS methods, their results overall did an excellent job predicting turbulence levels in the separated region. Although even at 200 million gridpoints the DNS simulation was still under-resolved at this Reynolds number, these computations demonstrated that this method can be a useful tool when looking for insight into the flow physics.

Other blended RANS-LES and LES papers have been published^{112, 113, 133, 145–147} for the hump model. Although the various results differed from each other to some degree because of different subgrid models (or lack thereof for implicit LES), different grid resolutions, and other factors, by and large these computations demonstrated the capability for using LES methodology in the separated region to improve predictions. In particular, by resolving many of the eddies in the separated shear layer region and capturing the dynamics of the large-scale motion, these methods produced increased levels of turbulence (and hence earlier reattachment), in better agreement with experiment than RANS models.

Although more costly to run than RANS, LES methods have become more affordable on today's computers, and they offer the prospect of obtaining a better understanding of the dynamics inherent in flow control problems involving separation. It remains to be seen whether their use will lead to improved RANS turbulence models for this class of flows.

D. Gaps and Needs

Section IV on flow control for high lift so far has discussed the results of two specific workshops in detail, in order to attempt to assess the state-of-the-art CFD capabilities in this area. Several gaps and needs have been identified, which are summarized here.

In terms of experiments, for CC flows many of the reference experiments are more than 30 years old, and there is a need for additional testing today. Efforts are currently under way in this area within NASA or funded by NASA. Of particular importance is that the experiments need to be designed specifically for the purpose of CFD validation. This requirement means several things. The simpler (lower down) the experiment is in the hierarchy of fig. 2, the easier it will be to perform unambiguous validation studies. Some important aspects include reporting clearly defined geometry and boundary conditions, particularly at and near any flow control slot. Measurements should include velocity profiles (in addition to the usual mass flow rate or C_μ), and a thorough assessment of two-dimensionality should be conducted for nominally 2-D configurations. This assessment should include angle-of-attack corrections due to wall and 3-D flow structure effects.

Multiple measurement techniques were shown in CFDVAL2004 to be valuable in quantifying experimental uncertainties. Some of the differences between techniques were surprisingly large, indicating that additional experimental research and testing is still needed for time-dependent flow control applications. This workshop also highlighted CFD's need for extremely detailed boundary condition information at and near flow control slots or openings; for unsteady applications, this type of information can be very challenging to acquire.

In terms of CFD, a current problem identified was that codes and models can be difficult to compare because of potentially non-standard implementations of models. For example, many codes contain a turbulence model labeled as "SST," but there are several variations – both documented and undocumented – that have been used. These variations include differences in limiters, treatment of the source term (approximate vs. exact), boundary conditions, and viscous modeling (full, thin-layer, or neglect of cross-derivative terms). The Spalart-Allmaras model also has several published and unpublished variations in some of its constants and terms, and it is not always known which version has been coded. So a clear need exists for documentation and version control for turbulence models in wide use today. Somehow it must be made perfectly clear exactly what went into any given model, and how it was implemented. A solution may be as simple as producing a reference website or document listing in detail the models and their known variants,

along with a naming convention for each. Differences in boundary conditions would also have to be accounted for. Standardized code verification testing may also help to establish consistency in this area.

The hump model test case identified a definitive failing of all RANS turbulence models tested: turbulent shear stress was underpredicted in magnitude in the separated region, and as a consequence the flow reattached too far downstream. This failing identifies a clear gap in today's technology for this particular type of flow feature (which occurs for both steady and unsteady flows): there is a need for improved RANS turbulence modeling.

Wider usage of LES, blended RANS-LES (such as DES), and DNS simulations have emerged in recent years, applied to flow control problems. While these have shown some promise of improvements in prediction capability (although at a dramatic increase in cost over RANS), they also have shown variation and dependency on grid size and quality, inflow boundary conditions, and subgrid model. Usage of laminar N-S, under-resolved DNS, and implicit LES (LES with no explicit subgrid model) are also becoming more common, but the relationships and differences between the three methods rely solely on numerical considerations and can be difficult to elucidate. Much research is ongoing in all of these areas, and it is clear that it should continue. One thing that is not clear at this time is whether these advanced methods will prove to be useful in developing improved RANS turbulence models, or whether their primary benefit will come from being prediction tools in their own right.

Finally, the same final comment made at the end of section III.D applies here as well: there should be a significant, high-visibility focus on continued coordinated verification and validation workshops. They appear to be one of the most effective ways to determine the current status of state-of-the-art within the aerodynamics community, as well as to map future needs and directions.

V. Recommendations and Conclusions

This paper has attempted to assess the state-of-the-art CFD capabilities for subsonic fixed wing aircraft by looking at a few recent specific workshops and studies for transonic cruise through buffet onset and flow control for high lift applications. It is recognized that this approach is from a somewhat narrow perspective, and that many other broad categories exist within the context of subsonic fixed wing aerodynamics. The current examples were selected in part because they fall within the authors' expertise. However, it is believed that the issues covered still span a reasonable range, allowing us to gauge in a general sense some of the important needs in this discipline.

The general recommendations from this assessment are:

- Work to improve the processes for analyzing and reducing or eliminating error sources. This task can be divided into several sub-tasks.
 - Actively encourage and support a wide range of future CFD verification and validation workshops. In particular, by focusing on fundamental “unit problems” (that include physics that are also important for complete systems), there will be a greater likelihood of achieving success in isolating and fixing problems with models.
 - Continue research in verification and validation.
 - Publish and disseminate guides and “lessons learned” documents.
 - Continue research in automatic grid adaptation.
 - Improve grid generation processes and practices, particularly for creating appropriately-refined grid sequences for complex 3-D configurations and for unstructured grids.
 - Work to reduce experimental uncertainties in geometry and boundary conditions.
- Develop processes to reduce ambiguity in model implementation and reporting practices.
- Continue active research in LES, blended RANS-LES, and DNS methods, models, numerics, and applications. Support research that uses these methods to actively develop and improve RANS models.

Two recommendations that were more specific to the particular applications highlighted in this paper are:

- In experiments, encourage new research for the purpose of CFD validation of flow control applications, including circulation control, error analysis with multiple measurement techniques, and acquiring detailed flowfield boundary condition information near flow control slots.
- Improve RANS turbulence models to properly account for turbulent shear stress (currently underpredicted in magnitude) in the separation region of certain classes of flowfields.

As can be seen, most points given in the general recommendations focus on issues associated with improved error analysis and reduction of uncertainty and error. The reason for this focus is the fact that it is almost impossible to make firm conclusions regarding physical models if uncertainty and/or error are unknown and thrown into the mix. Countless studies – including those that were carefully and specifically designed to quantify uncertainty and error – have been confounded by ambiguity when trying to determine the cause of differences between multiple codes or between CFD and experiment. This situation has been improving, but clearly the currently-accepted practices are inadequate for the aerodynamics community as a whole. It is also noted that the general recommendations apply to more than subsonic fixed wing CFD. Although these recommendations arose from analysis of subsonic fixed wing workshops and studies, there is no doubt that they are also of key importance across the speed range and in other disciplines.

CFD use for subsonic fixed wing aircraft has come a long way in the last 20 years. RANS models and methods have improved to the point where the aircraft industry has incorporated them into their design processes, and trusts results to some degree for certain configurations and flow conditions. But many off-design conditions remain entirely untrusted, and still have to be analyzed and assessed on a case-by-case basis. Future improvements can most effectively be achieved through coordinated efforts. At the highest level of the complete system, uncertainty quantification is most difficult, so it would likely be impossible to achieve gains without a continued significant investment at the basic unit problem level as well.

References

- ¹Wahls, R. A., “The National Transonic Facility: A Research Retrospective,” AIAA Paper 2001-0754, 2001.
- ²Steering Committee for the Decadal Survey of Civil Aeronautics, “Decadal Survey of Civil Aeronautics: Foundation for the Future,” National Research Council of the National Academies, The National Academies Press, Washington, D.C., 2006.
- ³Roy, C. J. and Blottner, F. G., “Review and Assessment of Turbulence Models for Hypersonic Flows,” *Progress in Aerospace Sciences*, Vol. 42, No. 7-8, 2006, pp. 469–530.
- ⁴Gatski, T. B., “Constitutive Equations for Turbulent Flows,” *Theoretical and Computational Fluid Dynamics*, Vol. 18, No. 5, 2004, pp. 345–369.
- ⁵Spalart, P. R., “Strategies for Turbulence Modelling and Simulations,” *International Journal of Heat and Fluid Flow*, Vol. 21, 2000, pp. 252–263.
- ⁶Kral, L. D., “Recent Experience with Different Turbulence Models Applied to the Calculation of Flow over Aircraft Components,” *Progress in Aerospace Sciences*, Vol. 34, No. 7-8, 1998, pp. 481–541.
- ⁷Speziale, C. G., “Turbulence Modeling for Time-Dependent RANS and VLES: A Review,” *AIAA Journal*, Vol. 36, No. 2, 1998, pp. 173–184.
- ⁸Tulapurkara, E. G., “Turbulence Models for the Computation of Flow Past Airplanes,” *Progress in Aerospace Sciences*, Vol. 33, No. 1-2, 1997, pp. 71–165.
- ⁹Pope, S. B., “Ten Questions Concerning the Large-Eddy Simulation of Turbulent Flows,” *New Journal of Turbulence*, Vol. 6, No. 35, 2004, pp. 1–24.
- ¹⁰Piomelli, U. and Balaras, E., “Wall-Layer Models for Large-Eddy Simulations,” *Annual Review of Fluid Mechanics*, Vol. 34, 2002, pp. 349–374.
- ¹¹Lesieur, M. and Metais, O., “New Trends in Large-Eddy Simulations of Turbulence,” *Annual Review of Fluid Mechanics*, Vol. 28, 1996, pp. 45–82.
- ¹²Roache, P. J., *Verification and Validation in Computational Science and Engineering*, Hermosa Publishers, Albuquerque, NM, 1998.
- ¹³Anonymous, “Guide for the Verification and Validation of Computational Fluid Dynamics Simulations,” American Institute of Aeronautics and Astronautics, AIAA G-077-1998, 1998.
- ¹⁴Luckring, J. M., Hensch, M. J., and Morrison, J. H., “Uncertainty in Computational Aerodynamics,” AIAA Paper 2003-0409, January 2003.
- ¹⁵Roache, P. J., “Code Verification by the Method of Manufactured Solutions,” *ASME Journal of Fluids Engineering*, Vol. 124, March 2002, pp. 4–10.
- ¹⁶Roy, C. J., Smith, T. M., and Ober, C. C., “Verification of a Compressible CFD Code using the Method of Manufactured Solutions,” AIAA Paper 2002-3110, June 2002.
- ¹⁷Bond, R. B., Ober, C. C., Knupp, P. M., and Bova, S. W., “Manufactured Solution for Computational Fluid Dynamics Boundary Condition Verification,” *AIAA Journal*, Vol. 45, No. 9, 2007, pp. 2224–2236.
- ¹⁸Roy, C. J., Tendeau, E., Veluri, S. P., Rifki, R., Luke, E. A., and Hebert, S., “Verification of RANS Turbulence Models in Loci-CHEM using the Method of Manufactured Solutions,” AIAA Paper 2007-4203, June 2007.
- ¹⁹Mani, M., Winkler, C. M., Figher, M. S., and Mackie, S. A., “A Post Processing Approach for CFD Error Associated with Grid Resolution,” AIAA Paper 2007-0101, January 2007.
- ²⁰Shih, T. I.-P. and Qin, Y. C., “A Posteriori Method for Estimating and Correcting Grid-Induced Errors in CFD Solutions Part 1: Theory and Method,” AIAA Paper 2007-0100, January 2007.
- ²¹Roy, C. J., Raju, A., and Hopkins, M. M., “Estimation of Discretization Errors Using the Method of Nearby Problems,” *AIAA Journal*, Vol. 45, No. 6, 2007, pp. 1232–1243.
- ²²Pierce, N. A. and Giles, M. B., “Adjoint and Defect Error Bounding and Correction for Functional Estimates,” AIAA Paper 2003-3846, June 2003.
- ²³Diskin, B. and Thomas, J. L., “Accuracy Analysis for Mixed-Element Finite-Volume Discretization Schemes,” National Institute of Aerospace, NIA Report No. 2007-08, August 2007.

- ²⁴Park, M. A., "Adjoint-Based, Three-Dimensional Error Prediction and Grid Adaptation," *AIAA Journal*, Vol. 42, No. 9, 2004, pp. 1854–1862.
- ²⁵Lu, J. and Darmofal, D. L., "Adaptive Precision Methodology for Flow Optimization via Discretization and Iteration Error Control," AIAA Paper 2004-1096, January 2004.
- ²⁶Oberkampf, W. L. and Barone, M. F., "Measures of Agreement Between Computation and Experiment: Validation Metrics," AIAA Paper 2004-2626, June-July 2004.
- ²⁷Trucano, T. G., Pilch, M., and Oberkampf, W. L., "On the Role of Code Comparisons in Verification and Validation," Sandia Report SAND2003-2752, August 2003.
- ²⁸Jameson, A., "Re-Engineering the Design Process Through Computation," *Journal of Aircraft*, Vol. 36, No. 1, 1999, pp. 36–50.
- ²⁹Nielsen, E. J. and Park, M. A., "Using an Adjoint Approach to Eliminate Mesh Sensitivities in Computational Design," *AIAA Journal*, Vol. 44, No. 5, 2006, pp. 948–953.
- ³⁰Hosder, S., Walters, R. W., and Perez, R., "A Non-Intrusive Polynomial Chaos Method for Uncertainty Propagation in CFD Simulations," AIAA Paper 2006-0891, January 2006.
- ³¹Godfrey, A. G. and Cliff, E. M., "Sensitivity Equations for Turbulent Flows," AIAA Paper 2001-1060, January 2001.
- ³²Dowding, K. J. and Blackwell, B. F., "Sensitivity Analysis of Nonlinear Heat Conduction," *ASME Journal of Heat Transfer*, Vol. 123, Feb. 2001, pp. 1–10.
- ³³Helton, J. C. and Davis, F. J., "Latin Hypercube Sampling and the Propagation of Uncertainty in Analysis of Complex Systems," *Reliability Engineering and System Safety*, Vol. 81, 2003, pp. 23–69.
- ³⁴Eca, L., Hoekstra, M., and Roache, P. J., "Verification of Calculations: an Overview of the 2nd Lisbon Workshop," AIAA Paper 2007-4089, June 2007.
- ³⁵Anonymous, "Standard for Models and Simulations," NASA-STD-(I)-7009, December 2006.
- ³⁶Salas, M. D., "Some Observations on Grid Convergence," *Computers & Fluids*, Vol. 35, 2006, pp. 688–692.
- ³⁷Kleib, B. and Wood, B., "Computational Simulations and the Scientific Method," *Journal of Aerospace Computing, Information, and Communication*, Vol. 3, June 2006, pp. 244–250.
- ³⁸Hall, R. M., Woodson, S. H., and Chambers, J. R., "Accomplishments of the Abrupt-Wing-Stall Program," *Journal of Aircraft*, Vol. 42, No. 3, 2005, pp. 653–660.
- ³⁹Woodson, S. H., Green, B. E., Chung, J. J., Grove, D. V., Parikh, P. C., and Forsythe, J. R., "Recommendations for Computational-Fluid-Dynamics Procedures for Predicting Abrupt Wing Stall," *Journal of Aircraft*, Vol. 42, No. 3, 2005, pp. 627–633.
- ⁴⁰Stokesberry, D., "CFD Modeling of F/A-18E/F Abrupt Wing Stall – A Discussion of Lessons Learned," AIAA Paper 2001-2662, June 2001.
- ⁴¹Levy, D. W., Vassberg, J. C., Wahls, R. A., Zickuhr, T., Agrawal, S., Pirzadeh, S., and Hemsch, M. J., "Summary of Data from the First AIAA CFD Drag Prediction Workshop," *Journal of Aircraft*, Vol. 40, No. 5, 2003, pp. 875–882.
- ⁴²Laffin, K. R., Vassberg, J. C., Wahls, R. A., Morrison, J. H., Brodersen, O. P., Rakowitz, M., Tinoco, E. N., and Godard, J., "Summary of Data from the Second AIAA CFD Drag Prediction Workshop," *Journal of Aircraft*, Vol. 42, No. 5, 2005, pp. 1165–1178.
- ⁴³Vassberg, J. C., Tinoco, E. N., Mani, M., Brodersen, O. P., Eisfeld, B., Wahls, R. A., Morrison, J. H., Zickuhr, T., Laffin, K. R., Mavriplis, D. J., "Summary of the Third AIAA CFD Drag Prediction Workshop," AIAA Paper 2007-0260, January 2007.
- ⁴⁴Hemsch, M. J., "Statistical Analysis of CFD Solutions from the Drag Prediction Workshop," AIAA Paper 2002-0842, January 2002.
- ⁴⁵Hemsch, M. J. and Morrison, J. H., "Statistical Analysis of CFD Solutions from 2nd Drag Prediction Workshop," AIAA Paper 2004-0556, January 2004.
- ⁴⁶Morrison, J. H. and Hemsch, M. J., "Statistical Analysis of CFD Solutions from the Third AIAA Drag Prediction Workshop," AIAA Paper 2007-0254, January 2007.
- ⁴⁷Rakowitz, M., Eisfeld, B., Schwaborn, D., and Sutcliffe, M., "Structured and Unstructured Computations on the DLR-F4 Wing-Body Configuration," *Journal of Aircraft*, Vol. 40, No. 2, 2003, pp. 256–264.
- ⁴⁸Mavriplis, D. J. and Levy, D. W., "Transonic Drag Predictions Using an Unstructured Multigrid Solver," AIAA Paper 2002-0838, January 2002.
- ⁴⁹Vassberg, J. C., Buning, P. G., and Rumsey, C. L., "Drag Prediction for the DLR-F4 Wing/Body Using OVERFLOW and CFL3D on an Overset Mesh," AIAA Paper 2002-0840, January 2002.
- ⁵⁰Rumsey, C. L. and Biedron, R. T., "Computation of Flow Over a Drag Prediction Workshop Wing/Body Transport Configuration Using CFL3D," NASA/TM-2001-211262, December 2001.
- ⁵¹Pfeiffer, N., "Reflections on the Second Drag Prediction Workshop," AIAA Paper 2004-0557, January 2004.
- ⁵²Brodersen, O., Rakowitz, M., Amant, S., Larrieu, P., Destarac, D., and Sutcliffe, M., "Airbus, ONERA, and DLR Results from the Second AIAA Drag Prediction Workshop," *Journal of Aircraft*, Vol. 42, No. 4, 2005, pp. 932–940.
- ⁵³Langtry, R. B., Kuntz, M., and Menter, F., "Drag Prediction of Engine-Airframe Interference Effects with CFX-5," *Journal of Aircraft*, Vol. 42, No. 6, 2005, pp. 1523–1529.
- ⁵⁴Sclafani, A. J., DeHaan, M. A., and Vassberg, J. C., "OVERFLOW Drag Predictions for the DLR-F6 Transport Configuration: A DPW-II Case Study," AIAA Paper 2004-0393, January 2004.
- ⁵⁵Rumsey, C. L., Rivers, S. M., and Morrison, J. H., "Study of CFD Variation on Transport Configurations from the Second Drag-Prediction Workshop," *Computers & Fluids*, Vol. 34, 2005, pp. 785–816.
- ⁵⁶Wutzler, K., "Aircraft Drag Predictions Using COBALT," AIAA Paper 2004-0395, January 2004.
- ⁵⁷May, G., van der Weide, E., Jameson, A., and Shankaran, S., "Drag Prediction of the DLR-F6 Configuration," AIAA Paper 2004-0396, January 2004.
- ⁵⁸Kim, Y., Park, S., and Kwon, J., "Drag Prediction of DLR-F6 Using the Turbulent Navier-Stokes Calculation with Multigrid," AIAA Paper 2004-0397, January 2004.
- ⁵⁹Yamamoto, K., Ochi, A., Shima, E., and Takaki, R., "CFD Sensitivity to Drag Prediction on DLR-F6 Configuration by Structured Method and Unstructured Method," AIAA Paper 2004-0398, January 2004.
- ⁶⁰Tinoco, E. and Su, T., "Drag Prediction with the Zeus/CFL3D System," AIAA Paper 2004-0552, January 2004.

- ⁶¹Klausmeyer, S., "Drag, Lift, and Moment Estimates for Transonic Aircraft Using the Navier-Stokes Equations," AIAA Paper 2004-0553, January 2004.
- ⁶²Lee-Rausch, E., Frink, N., Milholen, W., and Mavriplis, D., "Transonic Drag Prediction Using Unstructured Grid Solvers," AIAA Paper 2004-0554, January 2004.
- ⁶³Tinoco, E. N., Winkler, C., Mani, M., and Venkatakrishnan, V., "Structured and Unstructured Solvers for the 3rd CFD Drag Prediction Workshop," AIAA Paper 2007-0255, January 2007.
- ⁶⁴Mavriplis, D. J., "Results from the 3rd Drag Prediction Workshop Using the NSU3D Unstructured Mesh Solver," AIAA Paper 2007-0256, January 2007.
- ⁶⁵Sclafani, A. J., Vassberg, J. C., Harrison, N. A., DeHaan, M. A., Rumsey, C. L., Rivers, S. M., and Morrison, J. H., "Drag Predictions for the DLR-F6 Wing/Body and DPW Wings Using CFL3D and OVERFLOW on an Overset Mesh," AIAA Paper 2007-0257, January 2007.
- ⁶⁶Murayama, M. and Yamamoto, K., "Comparison Study of Drag Prediction for the 3rd CFD Drag Prediction Workshop by Structured and Unstructured Mesh Method," AIAA Paper 2007-0258, January 2007.
- ⁶⁷Brodersen, O., Eisfeld, B., Raddatz, J., and Frohnappfel, P., "DLR Results from the Third AIAA CFD Drag Prediction Workshop," AIAA Paper 2007-0259, January 2007.
- ⁶⁸Sreenivas, K., Mitchell, B., Sawyer, S., Karman, S., Nichols, S., and Hyams, D., "Computational Prediction of Forces and Moments for Transport Aircraft," AIAA Paper 2007-1088, January 2007.
- ⁶⁹Eliasson, P. and Peng, S.-H., "Drag Prediction for the DLR-F6 Wing-Body Configuration Using the Edge Solver," AIAA Paper 2007-0897, January 2007.
- ⁷⁰Rumsey, C. L., Allison, D. O., Biedron, R. T., Buning, P. G., Gainer, T. G., Morrison, J. H., Rivers, S. M., Mysko, S. J., and Witkowski, D. P., "CFD Sensitivity Analysis of a Modern Civil Transport Near Buffet-Onset Conditions," NASA/TM-2001-211263, December 2001.
- ⁷¹Rumsey, C. L., Morrison, J. H., and Biedron, R. T., "CFD Variability for a Civil Transport Aircraft Near Buffet-Onset Conditions," NASA/TM-2003-212149, February 2003.
- ⁷²Spalart, P. R., and Allmaras, S. R., 1994, "A One-Equation Turbulence Model for Aerodynamic Flows," *La Recherche Aerospatiale*, No. 1, pp. 5–21; also AIAA Paper 92-0439, 1992.
- ⁷³Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598–1605.
- ⁷⁴Gad-el-hak, M., *Flow Control, Passive, Active, and Reactive Flow Management*, Cambridge University Press, Cambridge, UK, 2000.
- ⁷⁵Jones, G. S., Viken, S. A., Washburn, A. E., Jenkins, L. N., and Cagle, C. M., "An Active Flow Circulation Controlled Flap Concept for General Aviation Aircraft Applications," AIAA Paper 2002-3157, June 2002.
- ⁷⁶Collis, S. S., Joslin, R. D., Seifert, A., and Theofilis, V., "Issues in Active Flow Control: Theory, Control, Simulation, and Experiment," *Progress in Aerospace Sciences*, Vol. 40, 2004, pp. 237–289.
- ⁷⁷Tilman, C. P., Kimmel, R. L., Addington, G. A., and Myatt, J. H., "Flow Control Research and Applications at the AFRL's Air Vehicle Directorate," AIAA Paper 2004-2622, June-July 2004.
- ⁷⁸Joslin, R. D., Thomas, R. H., and Choudhari, M. M., "Synergism of Flow and Noise Control Technologies," *Progress in Aerospace Sciences*, Vol. 41, 2005, pp. 363–417.
- ⁷⁹Paterson, E. G. and Baker, W. J., "Simulation of Steady Circulation Control for Marine-Vehicle Control Surfaces," AIAA Paper 2004-0748, January 2004.
- ⁸⁰Chang, P. A. III, Slomski, J., Marino, T., and Ebert, M. P., "Numerical Simulation of Two- and Three-Dimensional Circulation Control Problems," AIAA Paper 2005-0080, January 2005.
- ⁸¹Rumsey, C. L. and Ying, S. X., "Prediction of High Lift: Review of Present CFD Capability," *Progress in Aerospace Sciences*, Vol. 38, 2002, pp. 145–180.
- ⁸²Haase, W., Chaput, E., Elsholz, E., Leschziner, M. A., and Muller, U. R. (eds), *ECARP - European Computational Aerodynamics Research Project: Validation of CFD Codes and Assessment of Turbulence Models*, Vieweg, Braunschweig, 1997.
- ⁸³Jacot, D. and Mabe, J., "Boeing Active Flow Control System for the V-22," AIAA Paper 2000-2473, June 2000.
- ⁸⁴Jones, G. S., and Joslin, R. D. (ed.), *Proceedings of the 2004 NASA/ONR Circulation Control Workshop*, NASA/CP-2005-213509/PT1 and 2, June 2005.
- ⁸⁵Abramson, J., "Two-Dimensional Subsonic Wind Tunnel Evaluation of Two Related Cambered 15-Percent Circulation Control Airfoils," DTNSRDC ASed-373, September 1977.
- ⁸⁶Abramson, J. and Rogers, E. O., "High-Speed Characteristics of Circulation Control Airfoils," AIAA Paper 83-0265, January 1983.
- ⁸⁷Fasel, H. F., Gross, A., and Wernz, S., "Investigation of Turbulent Coanda Wall Jets Using DNS and RANS," *Applications of Circulation Control Technology*, edited by R. D. Joslin and G. S. Jones, *Progress in Aeronautics and Astronautics Series*, Vol. 214, F. K. Lee, editor-in-chief, AIAA, Reston VA, 2006, pp. 401–420.
- ⁸⁸Paterson, E. G. and Baker, W. J., "RANS and Detached-Eddy Simulation of the NCCR Airfoil," *Applications of Circulation Control Technology*, edited by R. D. Joslin and G. S. Jones, *Progress in Aeronautics and Astronautics Series*, Vol. 214, F. K. Lee, editor-in-chief, AIAA, Reston VA, 2006, pp. 421–444.
- ⁸⁹Chang, P. A. III, Slomski, J., Marino, T., Ebert, M. P., Abramson, J., "Full Reynolds-Stress Modeling of Circulation Control Airfoils," *Applications of Circulation Control Technology*, edited by R. D. Joslin and G. S. Jones, *Progress in Aeronautics and Astronautics Series*, Vol. 214, F. K. Lee, editor-in-chief, AIAA, Reston VA, 2006, pp. 445–466.
- ⁹⁰Swanson, R. C., Rumsey, C. L., and Anders, S. G., "Aspects of Numerical Simulation of Circulation Control Airfoils," *Applications of Circulation Control Technology*, edited by R. D. Joslin and G. S. Jones, *Progress in Aeronautics and Astronautics Series*, Vol. 214, F. K. Lee, editor-in-chief, AIAA, Reston VA, 2006, pp. 469–498.
- ⁹¹McGowan, G., Gopalarathnam, A., Xiao, X., and Hassan, H., "Role of Turbulence Modeling in Flow Prediction of Circulation Control Airfoils," *Applications of Circulation Control Technology*, edited by R. D. Joslin and G. S. Jones, *Progress in Aeronautics and Astronautics Series*, Vol. 214, F. K. Lee, editor-in-chief, AIAA, Reston VA, 2006, pp. 490–510.
- ⁹²Baker, W. J. and Paterson, E. G., "Simulation of Steady Circulation Control for the General Aviation Circulation Control (GACC) Wing," *Applications of Circulation Control Technology*, edited by R. D. Joslin and G. S. Jones, *Progress in Aeronautics and Astronautics Series*, Vol. 214, F. K. Lee, editor-in-chief, AIAA, Reston VA, 2006, pp. 513–537.

- ⁹³McGowan, G. and Gopalathnam, A., "Computational Study of a Circulation Control Airfoil Using FLUENT," *Applications of Circulation Control Technology*, edited by R. D. Joslin and G. S. Jones, *Progress in Aeronautics and Astronautics Series*, Vol. 214, F. K. Lee, editor-in-chief, AIAA, Reston VA, 2006, pp. 539–554.
- ⁹⁴Swanson, R. C., Rumsey, C. L., and Anders, S. G., "Progress Towards Computational Method for Circulation Control Airfoils," AIAA Paper 2005-0089, January 2005.
- ⁹⁵Swanson, R. C. and Rumsey, C. L., "Numerical Issues for Circulation Control Calculations," AIAA Paper 2006-3008, June 2006.
- ⁹⁶Zacharos, A. and Kontis, K., "Numerical Studies on Active Flow Circulation Controlled Flap Concept for Aeronautical Applications," AIAA Paper 2004-2314, June-July 2004.
- ⁹⁷Slomski, J. F., Gorski, J. J., Miller, R. W., and Marino, T. A., "Numerical Simulation of Circulation Control Airfoils as Affected by Different Turbulence Models," AIAA Paper 2002-0851, January 2002.
- ⁹⁸Spalart, P. R. and Shur, M., "On the Sensitization of Turbulence Models to Rotation and Curvature," *Aerospace Science and Technology*, Vol. 1, No. 5, 1997, pp. 297–302.
- ⁹⁹Warren, E. S. and Hassan, H. A., "Transition Closure Model for Predicting Transition Onset," *Journal of Aircraft*, Vol. 35, No. 5, 1998, pp. 769–775.
- ¹⁰⁰Rumsey, C. L. and Gatski, T. B., 2003, "Summary of EASM Turbulence Models in CFL3D with Validation Test Cases," NASA/TM-2003-212431.
- ¹⁰¹Launder, B., Reece, G., and Rodi, W., "Progress in the Development of a Reynolds-Stress Turbulence Closure," *Journal of Fluid Mechanics*, Vol. 68, No. 3, 1975, pp. 537–566.
- ¹⁰²Slomski, J. F., Chang, P. A., and Arunajatesan, S., "Large Eddy Simulation of a Circulation Control Airfoil," AIAA Paper 2006-3011, June 2006.
- ¹⁰³Rumsey, C. L., Gatski, T. B., Sellers, W. L., III, Vatsa, V. N., and Viken, S. A., "Summary of the 2004 Computational Fluid Dynamics Validation Workshop on Synthetic Jets," *AIAA Journal*, Vol. 44, No. 2, 2006, pp. 194–207.
- ¹⁰⁴Yamaleev, N. K. and Carpenter, M. H., "Quasi-one-Dimensional Model for Realistic Three-Dimensional Synthetic Jet Actuators," *AIAA Journal*, Vol. 44, No. 2, 2006, pp. 208–216.
- ¹⁰⁵Vatsa, V. N. and Turkel, E., "Simulation of Synthetic Jets Using Unsteady Reynolds-Averaged Navier-Stokes Equations," *AIAA Journal*, Vol. 44, No. 2, 2006, pp. 217–224.
- ¹⁰⁶Carpy, S. and Manceau, R., "Turbulence Modelling of Statistically Periodic Flows: Synthetic Jet into Quiescent Air," *Int. Journal of Heat and Fluid Flow*, Vol. 27, No. 5, 2006, pp. 756–767.
- ¹⁰⁷Cui, J. and Agarwal, R. K., "Three-Dimensional Computation of a Synthetic Jet in Quiescent Air," *AIAA Journal*, Vol. 44, No. 12, 2006, pp. 2857–2865.
- ¹⁰⁸Dandois, J., Garnier, E., and Sagaut, P., "Unsteady Simulation of Synthetic Jet in a Crossflow," *AIAA Journal*, Vol. 44, No. 2, 2006, pp. 225–238.
- ¹⁰⁹Morgan, P. E., Rizzetta, D. P., and Visbal, M. R., "High-Order Numerical Simulation of Turbulent Flow over a Wall-Mounted Hump," *AIAA Journal*, Vol. 44, No. 2, 2006, pp. 239–251.
- ¹¹⁰Krishnan, V., Squires, K. D., and Forsythe, J. R., "Prediction of Separated Flow Characteristics over a Hump," *AIAA Journal*, Vol. 44, No. 2, 2006, pp. 252–262.
- ¹¹¹Postl, D. and Fasel, H. F., "Direct Numerical Simulation of Turbulent Flow Separation from a Wall-Mounted Hump," *AIAA Journal*, Vol. 44, No. 2, 2006, pp. 263–272.
- ¹¹²Saric, S., Jakirlic, S., Djugum, A., and Tropea, C., "Computational Analysis of a Locally Forced Flow over a Wall-Mounted Hump at High-Re Number," *Int. Journal of Heat and Fluid Flow*, Vol. 27, No. 4, 2006, pp. 707–720.
- ¹¹³You, D., Wang, M., and Moin, P., "Large-Eddy Simulation of Flow over a Wall-Mounted Hump with Separation Control," *AIAA Journal*, Vol. 44, No. 11, 2006, pp. 2571–2577.
- ¹¹⁴Kotapati, R. B., Mittal, R., Cattafesta, L. N. III, "Numerical Study of a Transitional Synthetic Jet in Quiescent External Flow," *Journal of Fluid Mechanics*, Vol. 581, 2007, pp. 287–321.
- ¹¹⁵Yao, C.-S., Chen, F. J., and Neuhart, D., "Synthetic Jet Flowfield Database for Computational Fluid Dynamics Validation," *AIAA Journal*, Vol. 44, No. 12, 2006, pp. 3153–3157.
- ¹¹⁶Schaeffler, N. W. and Jenkins, L. N., "Isolated Synthetic Jet in Crossflow: Experimental Protocols for a Validation Dataset," *AIAA Journal*, Vol. 44, No. 12, 2006, pp. 2846–2856.
- ¹¹⁷Greenblatt, D., Paschal, K. B., Yao, C.-S., Harris, J., Schaeffler, N. W., and Washburn, A. E., "Experimental Investigation of Separation Control Part 1: Baseline and Steady Suction," *AIAA Journal*, Vol. 44, No. 12, 2006, pp. 2820–2830.
- ¹¹⁸Greenblatt, D., Paschal, K. B., Yao, C.-S., and Harris, J., "Experimental Investigation of Separation Control Part 2: Zero Mass-Efflux Oscillatory Blowing," *AIAA Journal*, Vol. 44, No. 12, 2006, pp. 2831–2845.
- ¹¹⁹Naughton, J. W., Viken, S., and Greenblatt, D., "Skin-Friction Measurements on the NASA Hump Model," *AIAA Journal*, Vol. 44, No. 6, 2006, pp. 1255–1265.
- ¹²⁰Vatsa, V. N. and Carpenter, M. H., "Higher Order Temporal Schemes with Error Controllers for Unsteady Navier-Stokes Equations," AIAA Paper 2005-5245, June 2005.
- ¹²¹Park, S. H., Yu, Y. H., and Byun, D. Y., "RANS Simulation of a Synthetic Jet in Quiescent Air," AIAA Paper 2007-1131, January 2007.
- ¹²²Kotapati, R. B. and Mittal, R., "Time-Accurate Three-Dimensional Simulations of Synthetic Jets in Quiescent Air," AIAA Paper 2005-0103, January 2005.
- ¹²³Xia, H. and Qin, N., "Comparison of Unsteady Laminar and DES Solutions of Synthetic Jet Flow," AIAA Paper 2006-3161, June 2006.
- ¹²⁴Rumsey, C. L., "Computation of a Synthetic Jet in a Turbulent Cross-Flow Boundary Layer," NASA/TM-2004-213273, October 2004.
- ¹²⁵Iaccarino, G., Marongiu, C., Catalano, P., and Amato, M., "RANS Modeling and Simulations of Synthetic Jets," AIAA Paper 2004-2223, June-July 2004.
- ¹²⁶Cui, J. and Agarwal, R. K., "3-D CFD Validation of an Axisymmetric Jet in Cross-Flow (NASA Langley Workshop Validation: Case 2)," AIAA Paper 2005-1112, January 2005.

- ¹²⁷Biedron, R., Vatsa, V., and Atkins, H., "Simulation of Unsteady Flows Using an Unstructured Navier-Stokes Solver on Moving and Stationary Grids," AIAA Paper 2005-5093, June 2005.
- ¹²⁸Rumsey, C. L., Schaeffler, N. W., Milanovic, I. M., and Zaman, K. B. M. Q., "Time-Accurate Computations of Isolated Circular Synthetic Jets in Crossflow," *Computers & Fluids*, Vol. 36, No. 6, 2007, pp. 1092–1105.
- ¹²⁹Milanovic, I. M., Zaman, K. B. M. Q., and Rumsey, C. L., "An Isolated Circular Synthetic Jet in Crossflow at Low Momentum-Flux Ratio," AIAA Paper 2005-1110, January 2005.
- ¹³⁰Rizzetta, D. P., Visbal, M. R., and Stanek, M. J., "Numerical Investigation of Synthetic-Jet Flowfields," *AIAA Journal*, Vol. 37, No. 8, 1999, pp. 919–927.
- ¹³¹Seifert, A. and Pack, L. G., "Active Flow Separation Control on Wall-Mounted Hump at High Reynolds Numbers," *AIAA Journal*, Vol. 40, No. 7, 2002, pp. 1363–1372.
- ¹³²Batten, P., Goldberg, U., and Chakravarthy, S., "Interfacing Statistical Turbulence Closures with Large-Eddy Simulation," *AIAA Journal*, Vol. 42, No. 3, 2004, pp. 485–492.
- ¹³³Israel, D. M., Postl, D., and Fasel, H. F., "A Flow Simulation Methodology for Analysis of Coherent Structures and Flow Control," AIAA Paper 2004-2225, June-July 2004.
- ¹³⁴Durbin, P. A., "Separated Flow Computations with the $k-\varepsilon-v^2$ Model," *AIAA Journal*, Vol. 33, No. 4, 1995, pp. 659–664.
- ¹³⁵Rumsey, C. L., "Effect of Turbulence Models on Two Massively-Separated Benchmark Flow Cases," NASA/TM-2003-212412, May 2003.
- ¹³⁶Driver, D. M. and Seegmiller, H. L., "Features of a Reattaching Turbulent Shear Layer in Divergent Channel Flow," *AIAA Journal*, Vol. 23, No. 2, 1985, pp. 163–171.
- ¹³⁷Thangam, S. and Speziale, C. G., "Turbulent Flow Past a Backward-Facing Step: A Critical Evaluation of Two-Equation Models," *AIAA Journal*, Vol. 30, No. 5, 1992, pp. 1314–1320.
- ¹³⁸Hanjalic, K. and Jakirlic, S., "Contribution Towards the Second-Moment Closure Modelling of Separating Turbulent Flows," *Computers & Fluids*, Vol. 27, No. 2, 1998, pp. 137–156.
- ¹³⁹Wilcox, D. C., *Turbulence Modeling for CFD*, Third Edition, DCW Industries, La Canada, 2006.
- ¹⁴⁰Balakumar, P., "Computations of Flow over a Hump Model Using Higher Order Method with Turbulence Modeling," AIAA Paper 2005-1270, January 2005.
- ¹⁴¹Capizzano, F., Catalano, P., Marongiu, C., and Vitagliano, P. L., "U-RANS Modelling of Turbulent Flows Controlled by Synthetic Jets," AIAA Paper 2005-5015, June 2005.
- ¹⁴²Rumsey, C., "Reynolds-Averaged Navier-Stokes Analysis of Zero Efflux Flow Control over a Hump Model," *Journal of Aircraft*, Vol. 44, No. 2, 2007, pp. 444–452.
- ¹⁴³Bettini, C. and Cravero, C., "Computational Analysis of Flow Separation Control for the Flow Over a Wall-Mounted Hump Using a Synthetic Jet," AIAA Paper 2007-0516, January 2007.
- ¹⁴⁴Rumsey, C. and Greenblatt, D., "Flow Control Predictions Using URANS Modeling: A Parametric Study," 5th International Symposium on Turbulence and Shear Flow Phenomena, TU Munich, Germany, August 27-29, 2007.
- ¹⁴⁵Hiller, S. J. and Seitz, P. A., "The Interaction Between a Fluidic Actuator and Main Flow Using SAS Turbulence Modeling," AIAA Paper 2006-3678, June 2006.
- ¹⁴⁶Morgan, P. E., Rizzetta, D. P., and Visbal, M. R., "Large-Eddy Simulation of Flow Over a Wall-Mounted Hump," AIAA Paper 2005-0484, January 2005.
- ¹⁴⁷Morgan, P. E., Rizzetta, D. P., and Visbal, M. R., "Large-Eddy Simulation of Separation Control for Flow Over a Wall-Mounted Hump," AIAA Paper 2005-5017, June 2005.

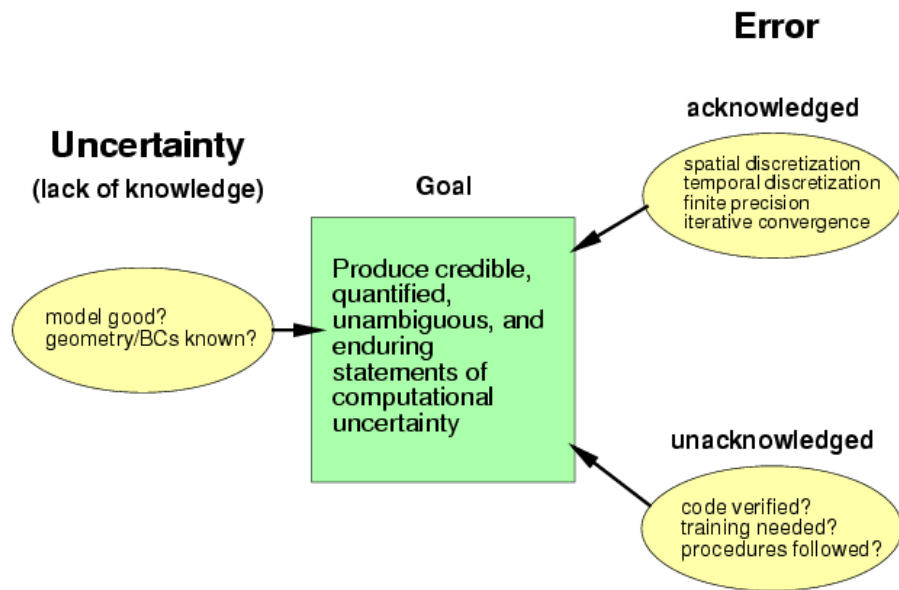


Figure 1. Graphic illustrating uncertainty and error contributions to loss in accuracy of CFD simulations.

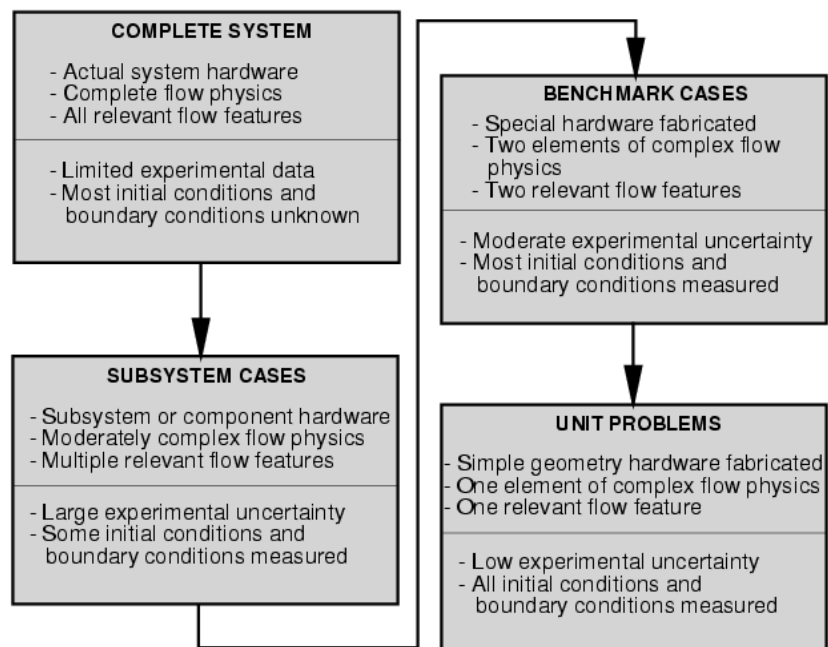


Figure 2. Characteristics of validation phases, from AIAA Guide for V&V.¹³

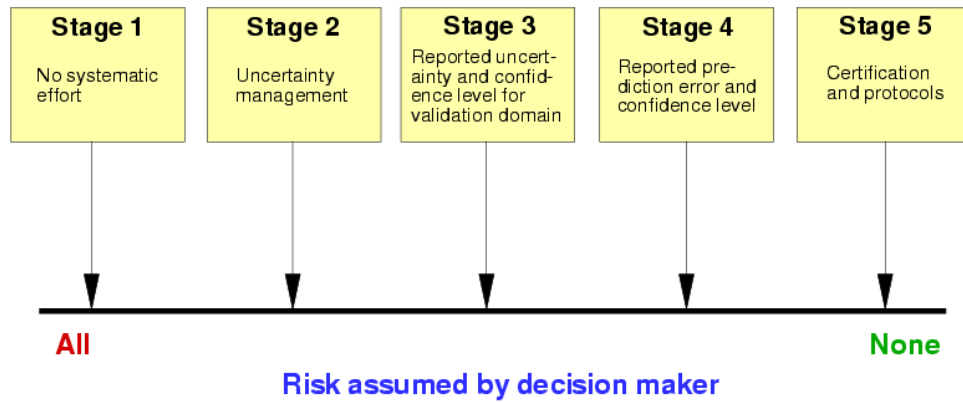


Figure 3. Mapping computational uncertainty stages onto decision-maker risk, from Luckring et al.¹⁴

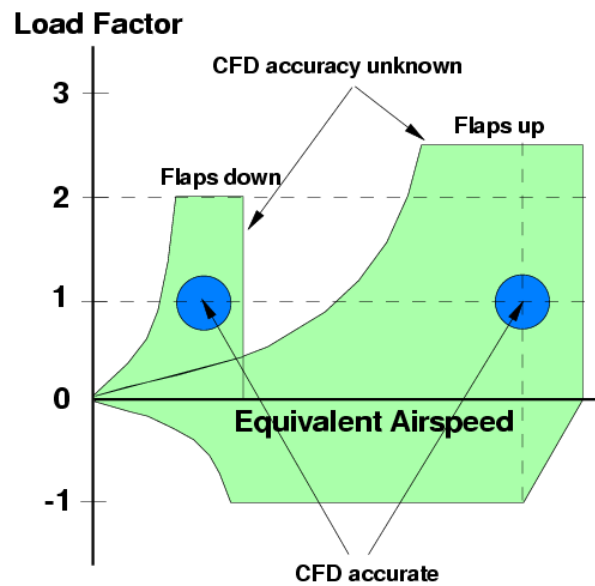


Figure 4. Graphic representing the flight envelope, from Luckring et al.¹⁴

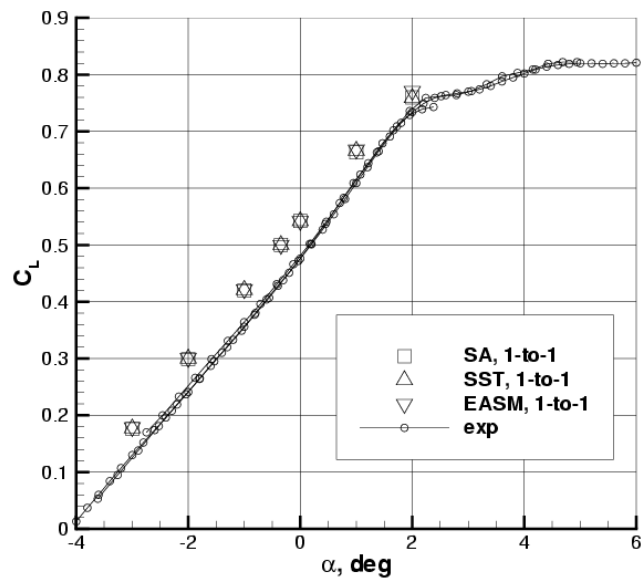


Figure 5. Typical lift curve predictions from DPW-I.

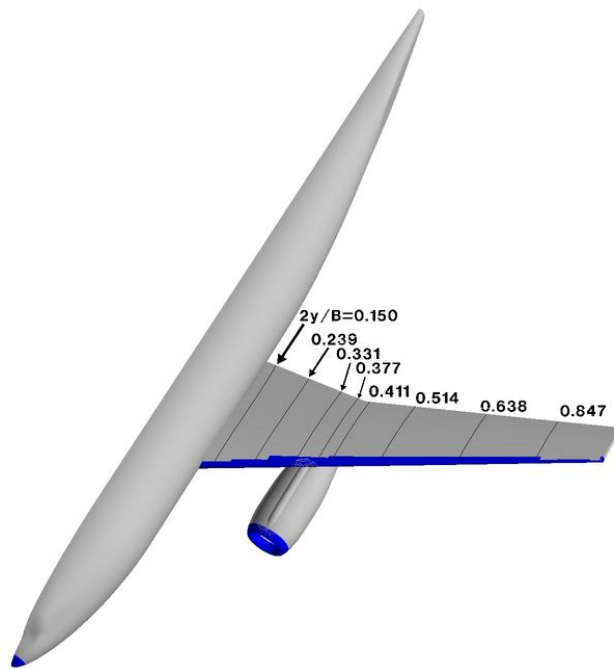


Figure 6. DLR-F6 wing-body-nacelle-pylon configuration from DPW-II, with laminar regions shaded.

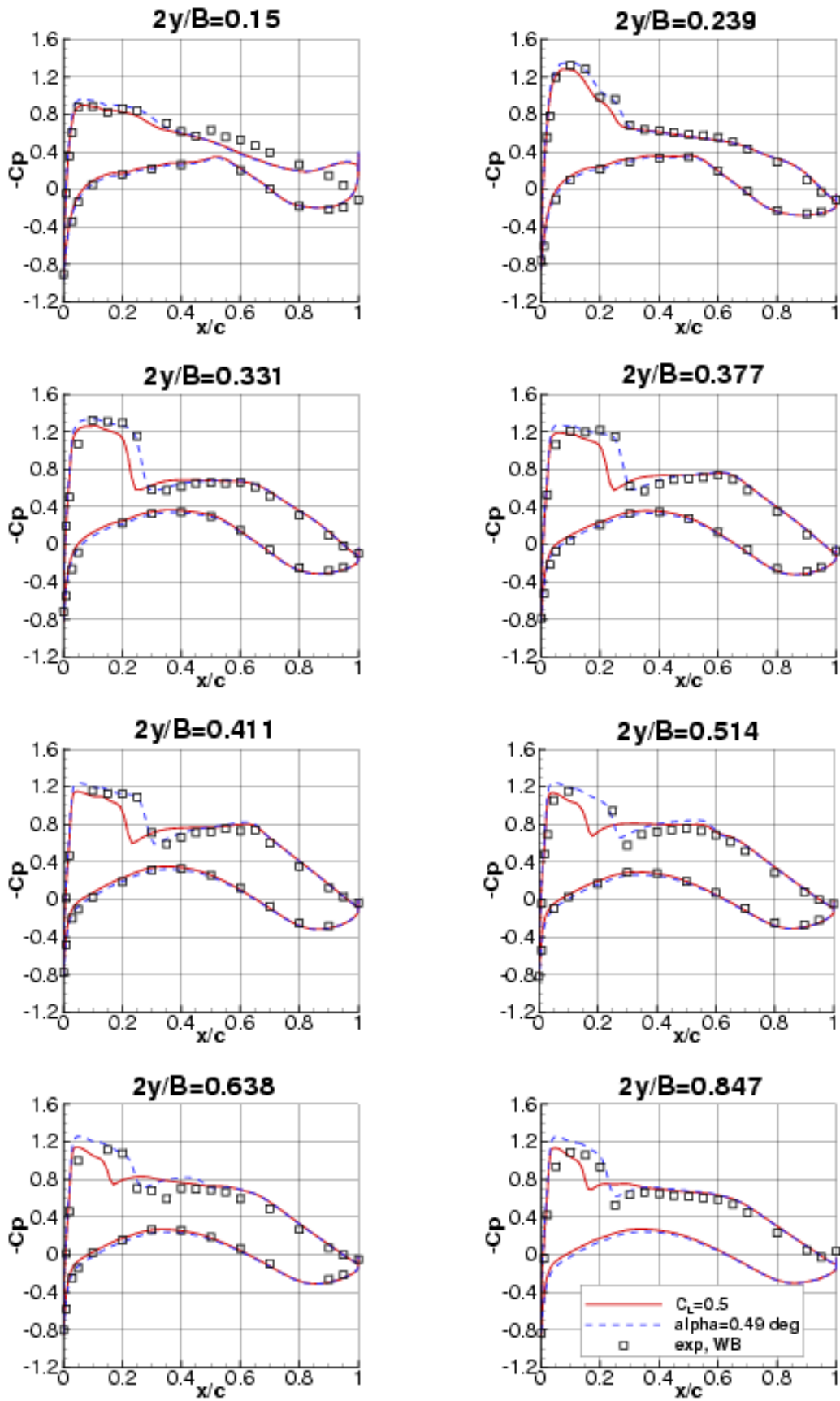


Figure 7. Typical surface pressure comparison for DLR-F6 wing-body, showing effect of matching lift vs. matching angle-of-attack.

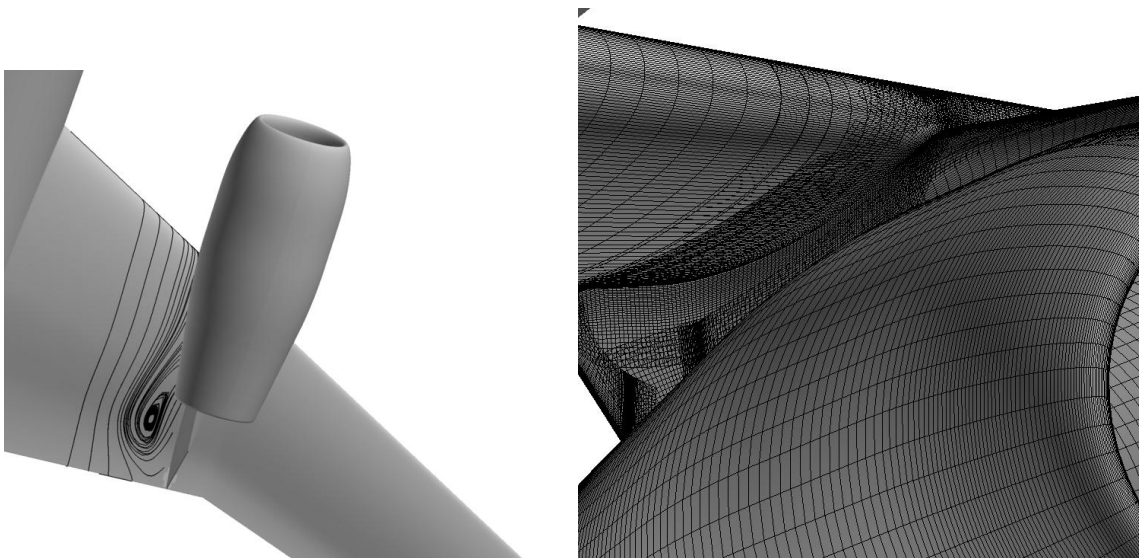


Figure 8. Streamlines (left) and corresponding overset grid (right) on the underside of the DLR-F6 wing-body-nacelle-pylon configuration.

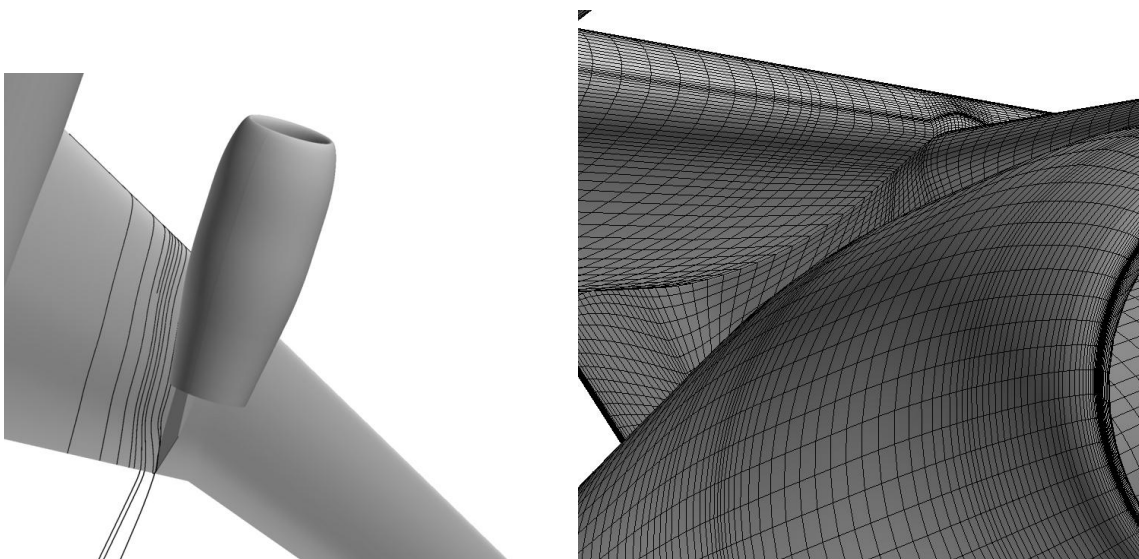


Figure 9. Streamlines (left) and corresponding 1-to-1 grid (right) on the underside of the DLR-F6 wing-body-nacelle-pylon configuration.

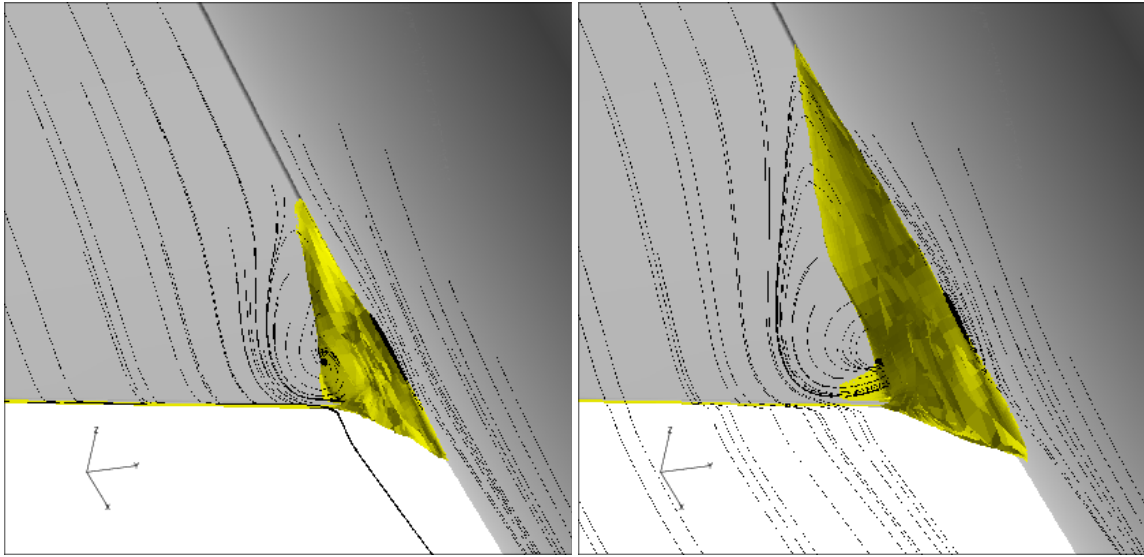


Figure 10. Example streamlines and bounding isosurface of reverse flow in the wing-root-juncture region (flow is from top to bottom in the figures); thin-layer Navier-Stokes (left), full Navier-Stokes (right).

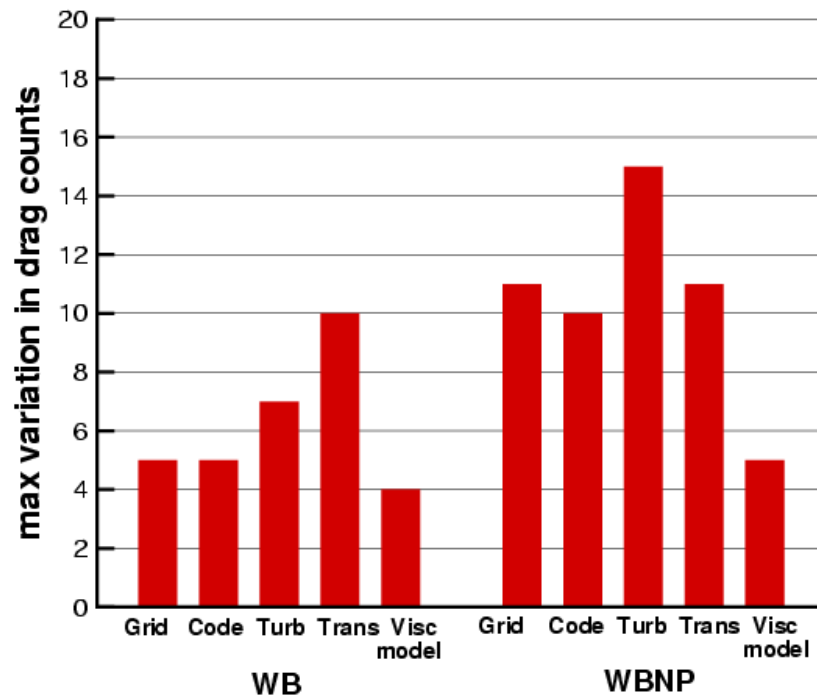


Figure 11. Summary of various effects on drag from DPW-II, from Rumsey et al.⁵⁵

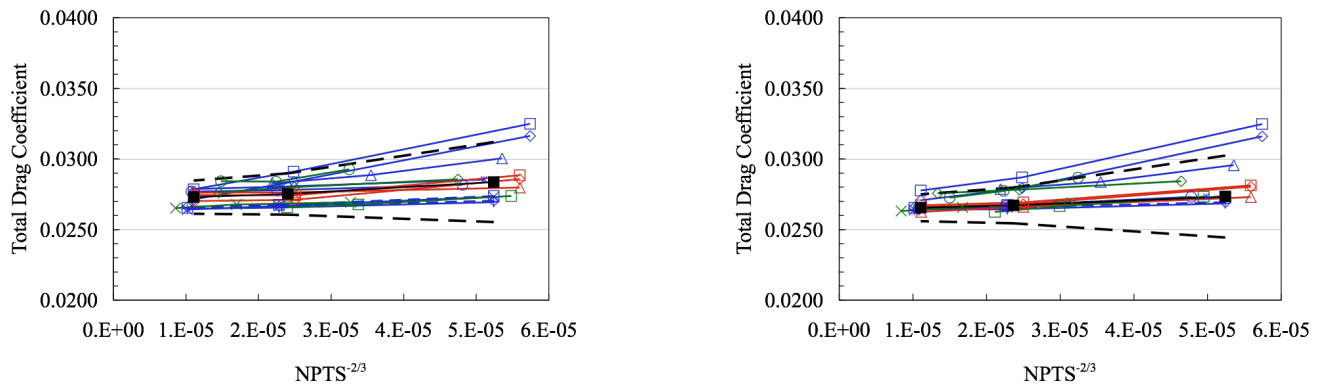


Figure 12. Total drag coefficient of all CFD core solutions from DPW-III as a function of grid size to the $-2/3$ power; DLR-F6 (left), FX2B (right) (from Morrison and Hensch⁴⁶).

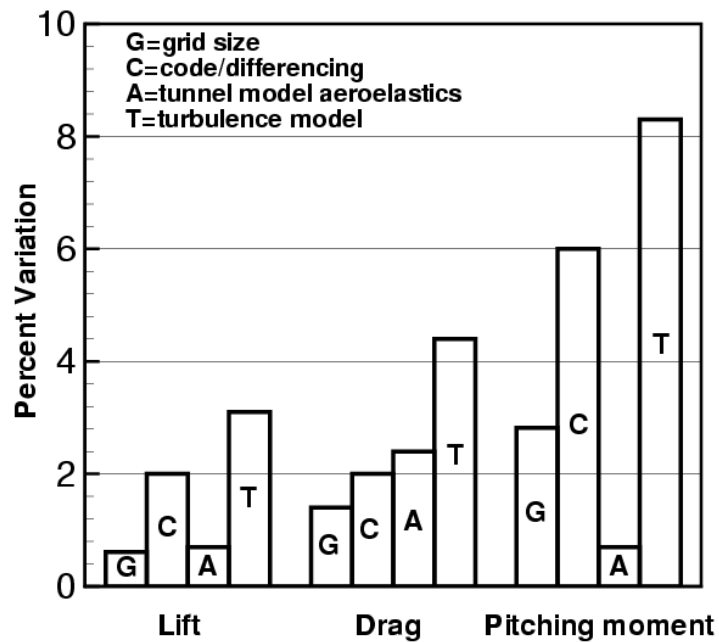


Figure 13. CFD variations for the buffet onset conditions in Rumsey et al.⁷⁰ (grid with 7.1 million points).

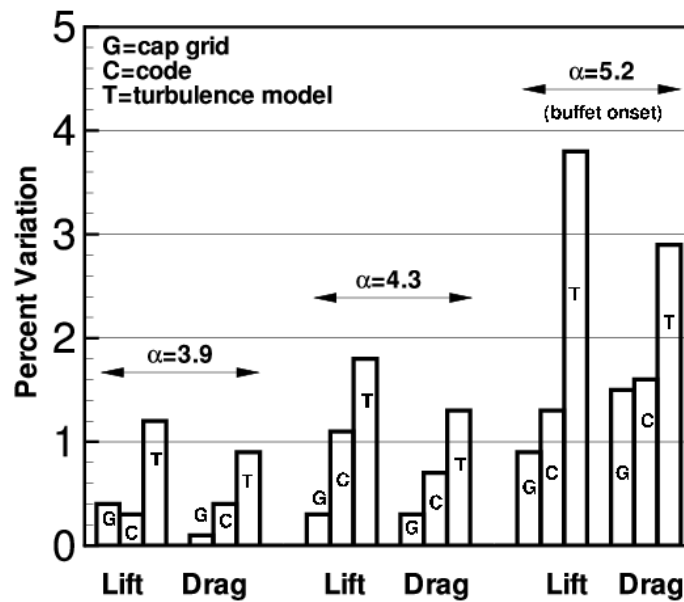


Figure 14. CFD variations for the separation and buffet onset conditions in Rumsey et al.⁷¹ (grid with 11.8 million points).

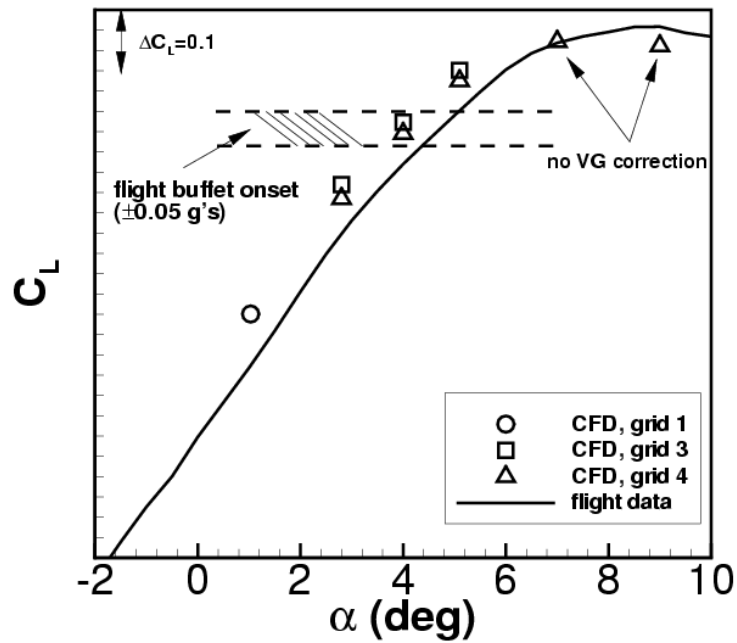


Figure 15. Comparison of computed lift with flight data, from Rumsey et al.⁷⁰

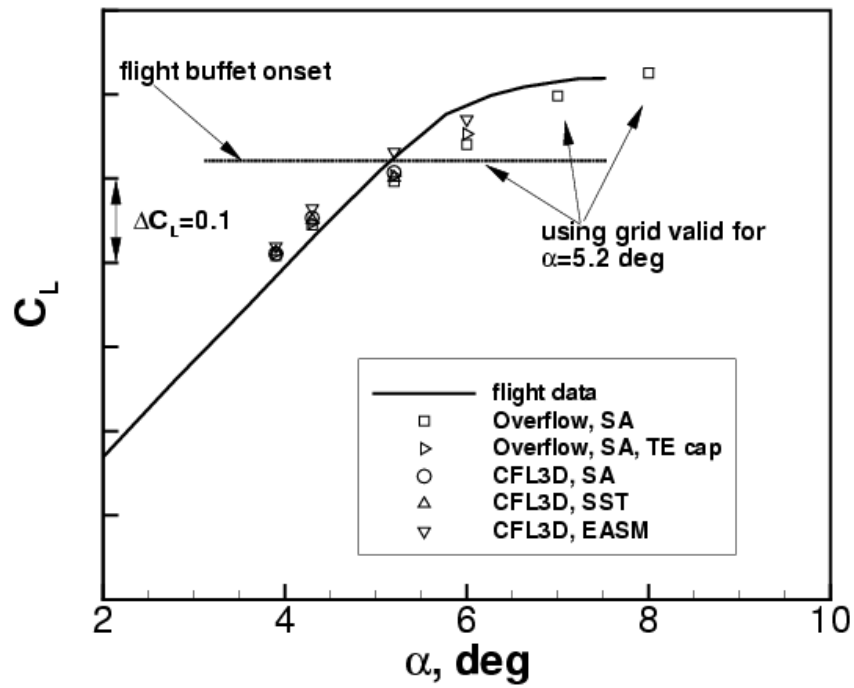


Figure 16. Comparison of computed lift with flight data, from Rumsey et al.⁷¹

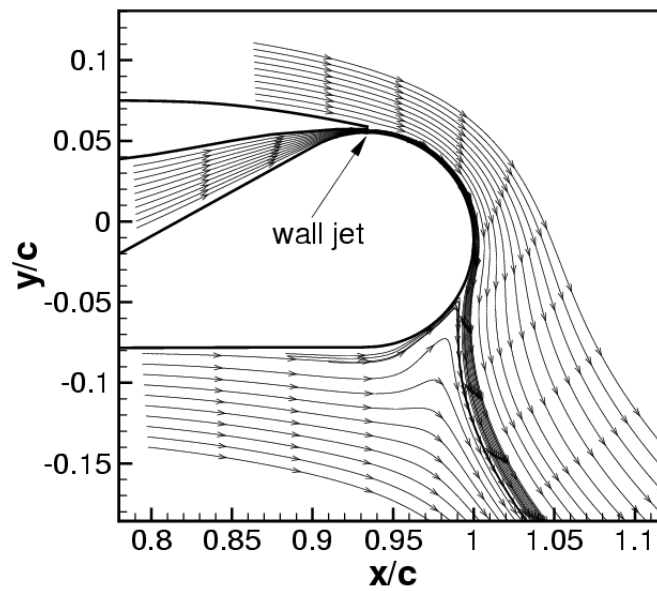


Figure 17. View of typical circulation control airfoil near trailing edge.

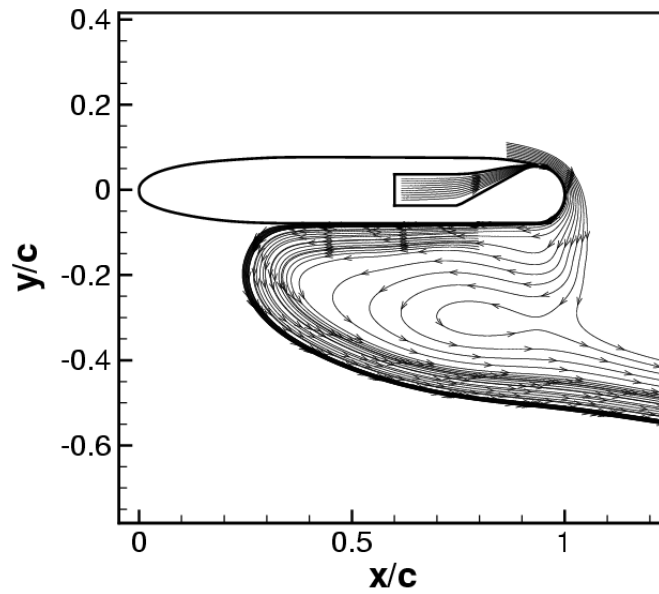


Figure 18. Example showing Coanda jet wrapping around the airfoil nonphysically far.

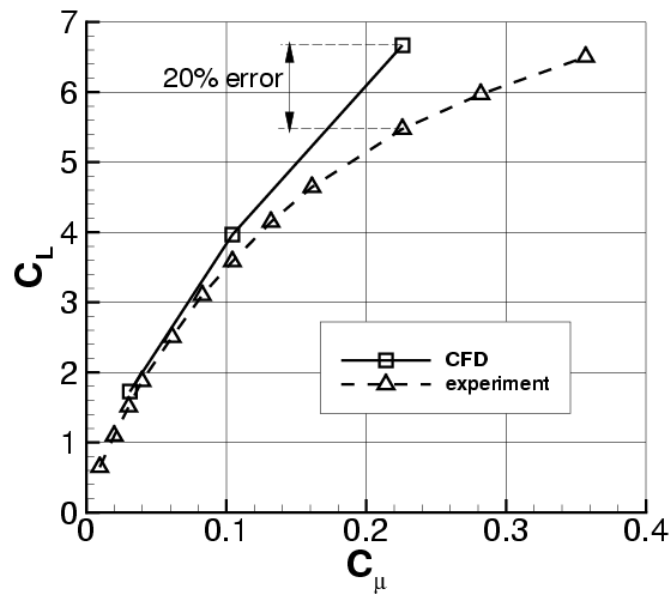


Figure 19. Example showing divergence of CFD prediction of lift for Coanda airfoil case with increasing jet blowing coefficient, from Swanson et al.⁹⁴

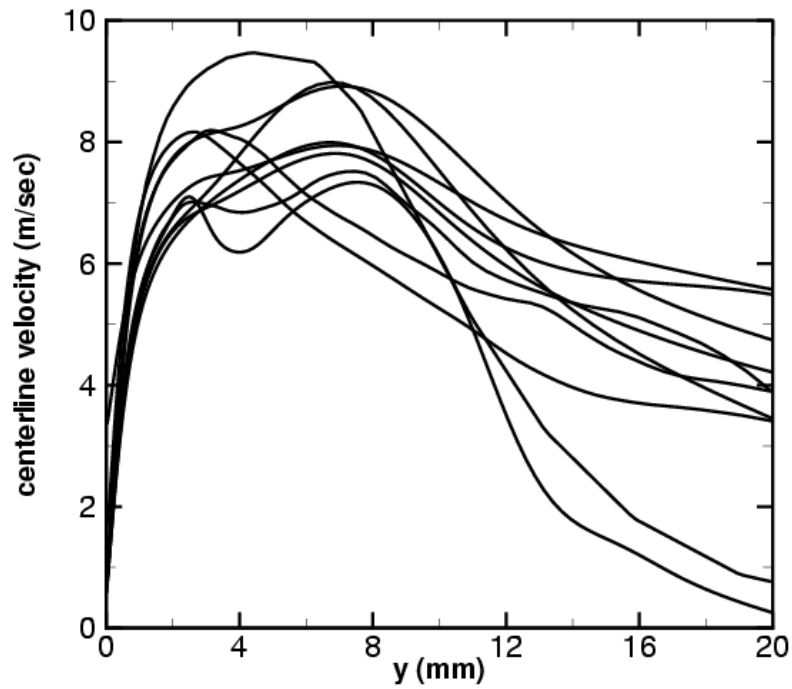


Figure 20. Time-averaged vertical velocity along the jet centerline for CFDVAL2004 case 1 (synthetic jet into quiescent flow), showing representative results from 10 different workshop participants.

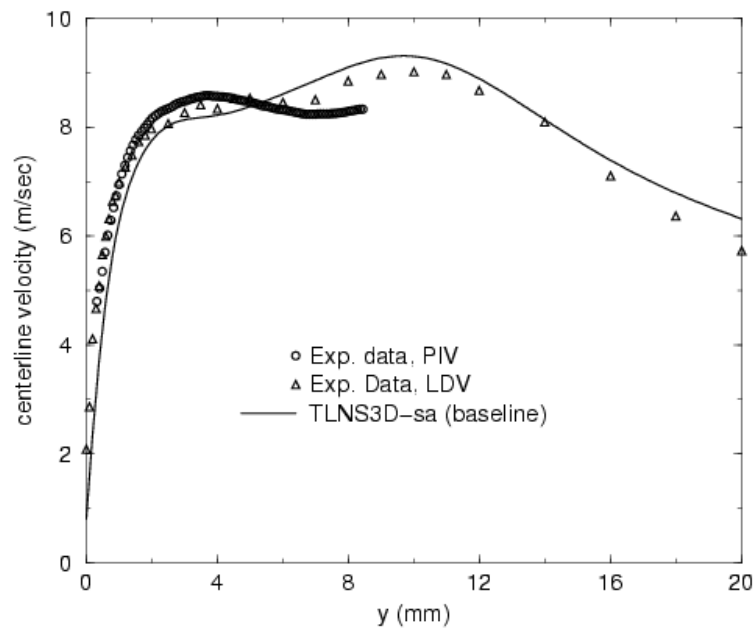


Figure 21. Time-averaged vertical velocity along the jet centerline for CFDVAL2004 case 1 (synthetic jet into quiescent flow), from computations of Vatsa and Turkel.¹⁰⁵

(a) comparison of CFD with experiment (LDV) (b) comparison of LDV with PIV measurements

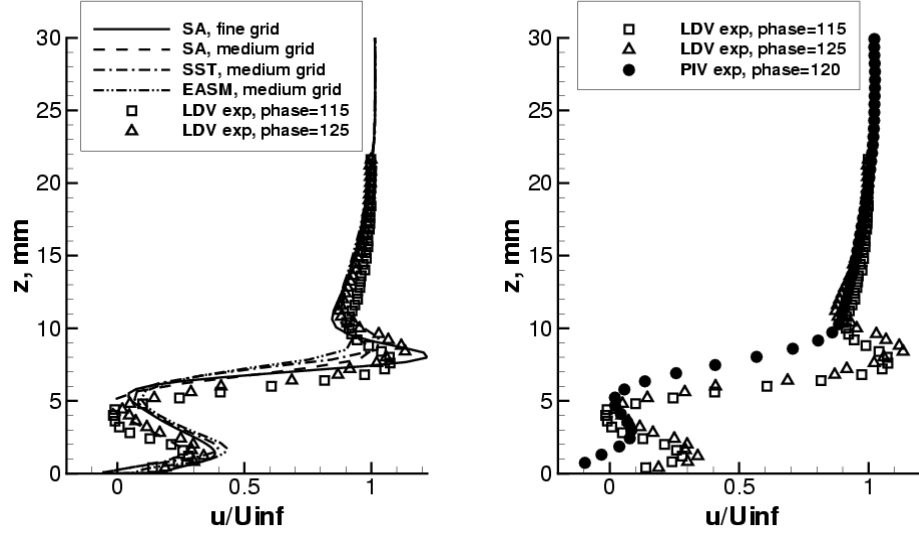


Figure 22. Profiles of phase-averaged u -velocity on wall 1-diameter downstream of orifice for CFDVAL2004 case 2 (synthetic jet into crossflow), at phase = 120°.

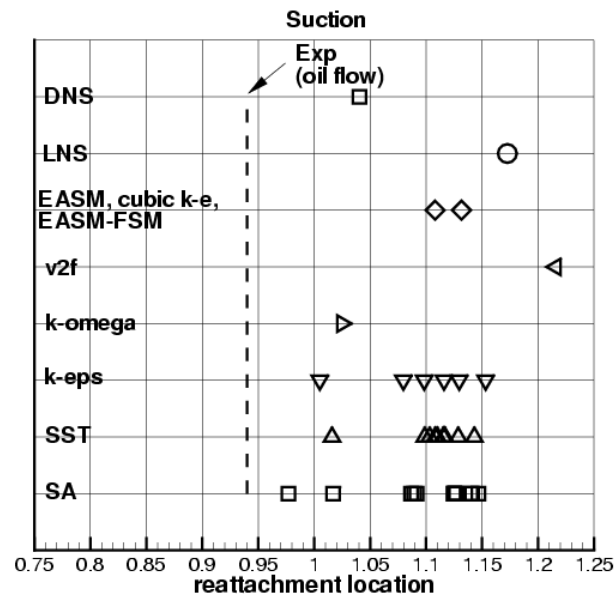


Figure 23. Reattachment location of CFDVAL2004 case 3 (hump model) workshop results with steady suction.

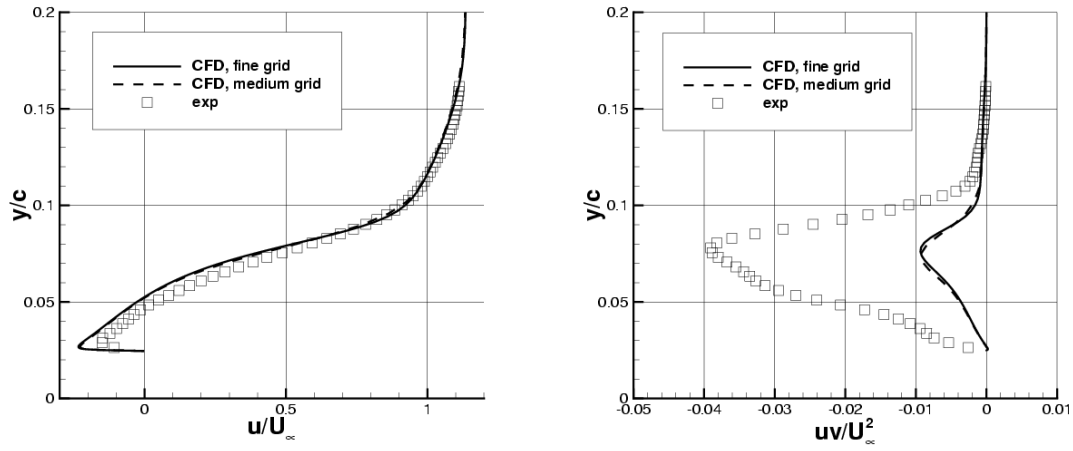


Figure 24. Typical velocity profiles (left) and turbulent shear stress profiles (right) at $x/c = 0.8$ in the separated region, for CFDVAL2004 case 3 (hump model).

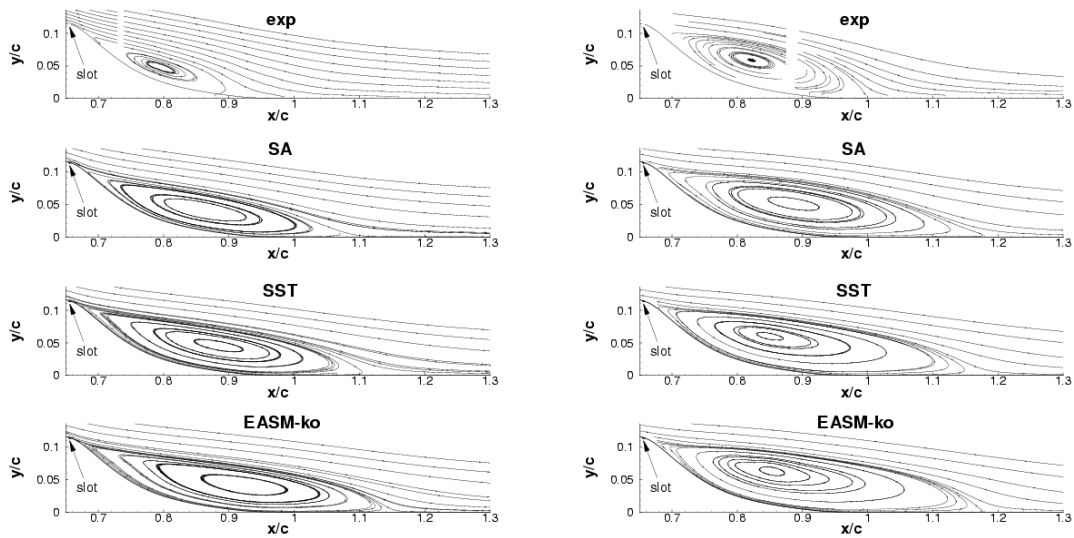


Figure 25. Typical streamlines for steady suction (left) and mean streamlines for oscillatory control (right) using three different RANS turbulence models, for CFDVAL2004 case 3 (hump model).

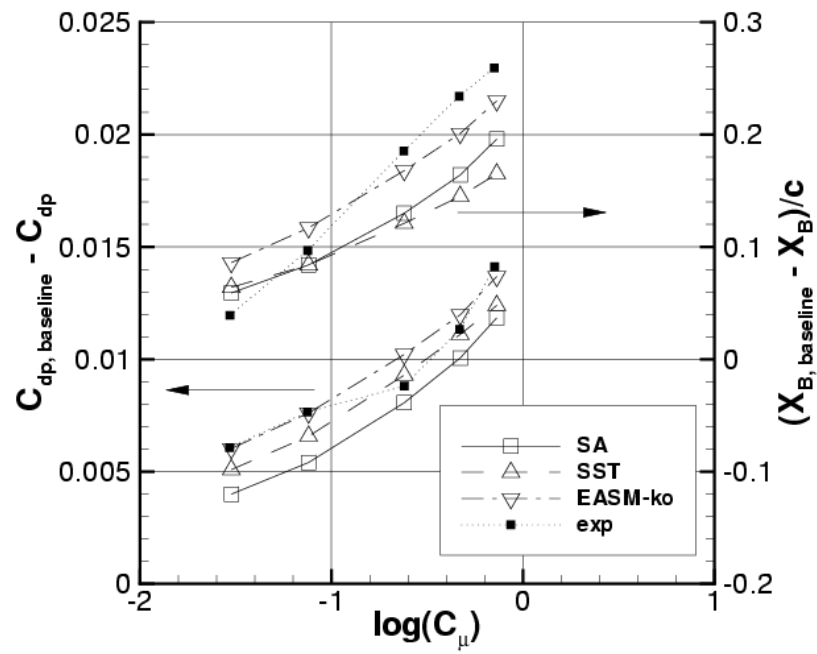


Figure 26. Bubble length and pressure drag coefficient relative to baseline (no control) as a function of C_μ for steady suction, for CFD-VAL2004 case 3 (hump model).

REPORT DOCUMENTATION PAGE					Form Approved OMB No. 0704-0188	
<p>The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.</p> <p>PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.</p>						
1. REPORT DATE (DD-MM-YYYY)		2. REPORT TYPE			3. DATES COVERED (From - To)	
01-06-2008		Technical Memorandum				
4. TITLE AND SUBTITLE Focused Assessment of State-of-the-Art CFD Capabilities for Prediction of Subsonic Fixed Wing Aircraft Aerodynamics				5a. CONTRACT NUMBER		
				5b. GRANT NUMBER		
				5c. PROGRAM ELEMENT NUMBER		
6. AUTHOR(S) Rumsey, Christopher L.; and Wahls, Richard A.				5d. PROJECT NUMBER		
				5e. TASK NUMBER		
				5f. WORK UNIT NUMBER 561581.02.08.07.20.02		
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NASA Langley Research Center Hampton, VA 23681-2199				8. PERFORMING ORGANIZATION REPORT NUMBER L-19484		
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001				10. SPONSOR/MONITOR'S ACRONYM(S) NASA		
				11. SPONSOR/MONITOR'S REPORT NUMBER(S) NASA/TM-2008-215318		
12. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category 01 Availability: NASA CASI (301) 621-0390						
13. SUPPLEMENTARY NOTES An electronic version can be found at http://ntrs.nasa.gov						
14. ABSTRACT Several recent workshops and studies are used to make an assessment of the current status of CFD for subsonic fixed wing aerodynamics. Uncertainty quantification plays a significant role in the assessment, so terms associated with verification and validation are given and some methodology and research areas are highlighted. For high-subsonic-speed cruise through buffet onset, the series of drag prediction workshops and NASA/Boeing buffet onset studies are described. For low-speed flow control for high lift, a circulation control workshop and a synthetic jet flow control workshop are described. Along with a few specific recommendations, gaps and needs identified through the workshops and studies are used to develop a list of broad recommendations to improve CFD capabilities and processes for this discipline in the future.						
15. SUBJECT TERMS CFD; Aerodynamics; Fixed-wing; Subsonic						
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON	
a. REPORT	b. ABSTRACT	c. THIS PAGE			STI Help Desk (email: help@sti.nasa.gov)	
U	U	U	UU	39	19b. TELEPHONE NUMBER (Include area code) (301) 621-0390	